A

Dissertation Report

on

CFD Modeling and Validation of a single slope Solar Still

Submitted in Partial Fulfillment of the Requirements for the Award of Degree of

Master of Technology

in

Energy Engineering

By

AMIT KUMAR

(2013PME5137)

Under the supervision of

Dr. Nirupam Rohatgi
Associate Professor
Department of Mechanical Engineering
M.N.I.T. Jaipur

DEPARTMENT OF MECHANICAL ENGINEERING

MALAVIYA NATIONAL INSTITUTE OF TECHNOLOGY, JAIPUR

JUNE 2015

DEPARTMENT OF MECHANICAL ENGINEERING
JAIPUR (RAJASTHAN)-302017
MALAVIYA NATIONAL INSTITUTE OF TECHNOLOGY

CERTIFICATE

This is certified that the dissertation report entitled **"CFD Modeling and Validation of a single slope Solar Still"** prepared by **Amit Kumar** (ID-2013PME5137), in the partial fulfillment of the award of the Degree **Master of Technology** in **Energy Engineering** of Malaviya National Institute of Technology Jaipur is a record of bona fide research work carried out by him under my supervision and is hereby approved for submission. The contents of this dissertation work, in full or in parts, have not been submitted to any other Institute or University for the award of any degree or diploma.

Date:

Place:

Dr. Nirupam Rohatgi
Associate Professor
Department of Mechanical Engineering
MNIT, Jaipur, India

DEPARTMENT OF MECHANICAL ENGINEERING

MALAVIYA NATIONAL INSTITUTE OF TECHNOLOGY

JAIPUR (RAJASTHAN)-302017

DECLARATION

I **Amit Kumar** hereby declare that the dissertation entitled **"CFD Modeling and Validation of a single slope Solar Still"** being submitted by me in partial fulfillment of the degree of **M. Tech (Energy Engineering)** is a research work carried out by me under the supervision of **Dr. Nirupam Rohatgi** and the contents of this dissertation work, in full or in parts, have not been submitted to any other Institute or University for the award of any degree or diploma. I also certify that no part of this dissertation work has been copied or borrowed from anyone else. In case any type of plagiarism is found out, I will be solely and completely responsible for it.

Date: AMIT KUMAR

M.Tech. (Energy Engg.)

Place: 2013PME5137

ACKNOWLEDGEMENT

It's an immense pleasure to express my sincere gratitude to my supervisor **Dr. Nirupam Rohatgi**, Department of Mechanical Engineering, for his constant encouragement and guidance from inception to completion of this dissertation work by taking interest and giving personal attention to the same. His valuable feedback, criticism and moral support have been a great source of inspiration for broadening my horizons in this area of research. In this respect I feel myself lucky to have him as my supervisor.

I would like to express my deepest gratitude to **Prof. Jyotirmay Mathur, Prof. Dilip Sharma, Dr. G.D. Aggarwal** and **Prof. S.L. Soni,** Department of Mechanical Engineering, Malaviya National Institute of Technology Jaipur, for their valuable guidance and support throughout my project work.

I am extremely thankful to **Prof. Rakesh Jain,** Head, Department of Mechanical Engineering, MNIT, Jaipur and all the faculty members for their motivation and moral support.

I feel indebted to all my **colleagues** for their constant help and moral support throughout my dissertation work. I also express my deepest gratitude to my **parents** for their blessings and affection, without which I would not be able to endure hard time and carry on with my studies.

(AMIT KUMAR)

ABSTRACT

Water is an essential requirement to sustain life. Although water is a renewable resource, yet most of the water present on earth is not suitable for direct usage. The world's supply of drinkable water is steadily decreasing, with depletion occurring most prominently due to rising population, Urbanization, Industrialization and environmental pollution. One third of world’s population live in countries with insufficient freshwater supply. Natural resources of water can only meet the fresh water demand up to the limited extent. In order to meet the ever increasing demand of potable water, purification of saline water with the help of effective water desalination method becomes necessary.

Solar still is a device which works on solar desalination method. It can solve the problem of potable water without using the high grade energy. It is potentially applicable to provide fresh water for remote areas where electrical energy is rather scarce. Solar still suffers greatly with the problem of its low productivity of drinkable water. Therefore, it is required to study the modeling and transport parameters for an efficient design.

In this work, a multi-phase three dimensional CFD model of a single slope solar still has been made. The model can simulate the temperatures at various points inside the still. The simulation results have been compared with the available experimental data. Within the scope of this study, simulation results are in good agreement with the experimental results. The effect of basin water depth on the distillation yield has also been investigated. It has a little effect on the productivity of solar still. On reducing the initial basin water quantity from 20 litres to 10 litres, the distillate output was increased by 5.13 %.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

A_{eff}	Effective basin Area of solar still [m^2]
C_p	Specific heat of water [kJ/kg K]
h_{cw}	Convective Heat transfer coefficient [W/m^2 K]
h_{ew}	Evaporative heat transfer coefficient [W/m^2 K]
h_{fg}	Latent heat of vaporization of water [kJ/kg]
I	Solar intensity [W/m^2]
K	Thermal conductivity [W/m K]
k	Turbulent kinetic energy [kJ/kg]
M	Mass of water in the basin [kg]
m	Cumulative distillate output [kg/m^2]
P_W	Saturated pressure at water temperature [kPa]
P_g	Saturated pressure at glass temperature [kPa]
Q	Heat transfer from various surfaces [kJ]
T	Temperature [K]
U	Heat loss coefficient [$W/m^2.K$]
V	Wind velocity [m/s]
μ	Dynamic viscosity [kg/m-s]
Tg	Glass temperature [K]
Tv	Vapor temperature
Tw	Water temperature
ε	K.E. dissipation rate
ω	Specific dissipation rate[s^{-1}]
∇	Gradient operator

CHAPTER 1
INTRODUCTION

1.1 Background

Water, Air and energy are the most essential things to sustain human life on Earth. Water and human life are two inseparable things. Most of the early civilizations of men are found to be developed at the river banks. Water plays a key role in the development and welfare of civilization. All ecosystems and every field of human activity directly or indirectly depend on clean water and that is why the availability of clean water is most important issue on International agenda.

Water is the precious gift from nature and is present in abundance. Over 70% of the Earth‘s surface is covered by water but most of it is not suitable for direct human consumption. Around 97% of the water on earth is in ocean, approximately 2% of the water is stored as ice in polar region and only 1% of total water is available for the need of humans, plants and animals which is in the form of rivers, lakes and underground reservoirs [4]. However this small fraction of available fresh water is adequate to support life and vegetation on earth but the ever increasing population and rapid industrial growth along with water pollution caused by ignorance of human being, led the world towards water scarcity.

The need for potable water is increasing due to the number of factors such as population growth, Urbanization, Industrialization, etc. As population of this planet has rapidly grown, we have increasingly tapped deeper into our planets fresh water resources [24]. The availability of water in sufficient quantities and quality is a challenge of significant importance in many regions as water is a scarce and unevenly distributed resource. Potable water is scarce in many areas of the world. More than 780 million people suffer from the absence of a reliable source of drinking water [16].According to World Health Organization (WHO) almost 1 billion people still lack the access to an improved water supply line. Contaminated water can transmit dangerous diseases such as diarrhea, cholera, dysentery, typhoid and polio.

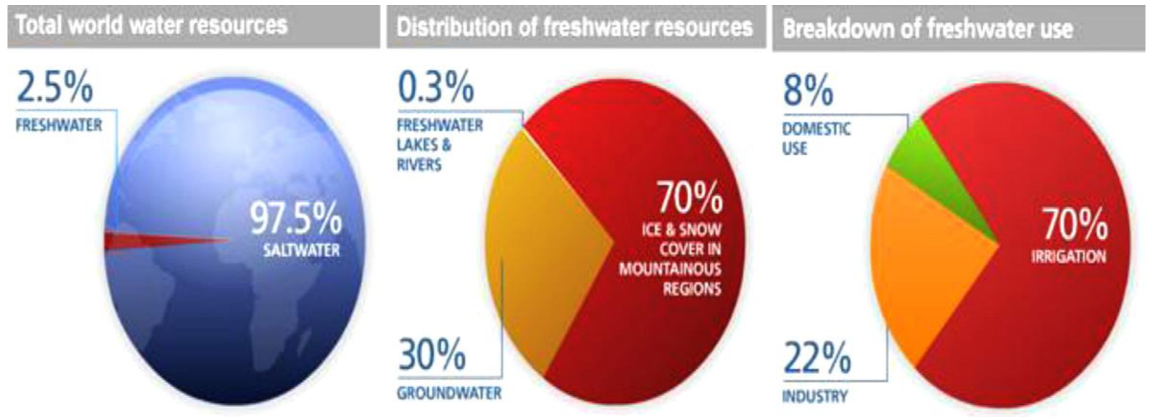

Figure 1.1 Distribution of world water resource [25]

It is anticipated that half of the world's population will be living in water stressed areas by 2025. Water pollution accounts for 80% of all infectious diseases. It has been anticipated that 4% of the global disease burden could be prevented only by improving water supply, sanitation, and hygiene.

Non-availability of potable water is one of the main problem faced by both the under developed and developing countries all over the world. The situation is getting worse by the day because of rapid industrial growth. Clearly, there is a worldwide potential need for applying conservative measures and effective management to save this valuable resource. Water desalination is an effective method to assist the ever increasing demand of potable water. According to the 22nd GWI/IDA Worldwide Desalting Plant Inventory, there are 14,451 desalination plants on-line and the global total capacity is approximately 60 million-m^3/day [23]. Desalination of water is an oldest technique to convert brackish water, river water or salty water into potable water. With the limited fresh water resources and ever increasing water demand, there is a potential need to improve as well as increase the water desalination plants in order to meet the potable water demand.

1.2 Water desalination methods

Desalination of seawater to produce clean water is the oldest method of water purification and it has emerged as one of the most sustainable solution to provide drinkable water in many countries. Water desalination improves the access to safe drinkable water which can result in multidimensional socio-economic effects such as improvement in health and hygiene, improvements in rural and urban livelihood, children’s attendance at school, psychological well-being and social interactions [27]. Desalination is a process in which saline water is separated into two parts using different forms of input energy such as fossil fuel, electricity or solar energy. One part that has a low concentration of dissolved salts can be used as drinkable water and the other one which has higher concentration of dissolved salts than the original feed water is generally discarded.

All over the world, the installed desalination capacity is increasing day by day because of growing clean water demand. Currently there are more than 14,000 desalination plants all over the world producing sixty million cubic meters of water per day, according to International Desalination Association (IDA). It is expected to be doubled by the end of year 2015. Since conventional desalination plants are operated with fossil fuel, they are becoming more expensive to run because of the rise in world energy price. Approximately, production of 1000 m^3 per day of fresh water requires 10,000 tonnes of oil per year which can be considered as highly significant energy consumption and it will increase day by day [23]. There are several methods available for the water desalination and most of them can be classified into two categories namely phase change thermal process and membrane process.

1.2.1 Thermal desalination

Thermal desalination works on the principle of phase change i.e. evaporation and condensation of water. Thermal desalination process is very simple. The brackish or saline water is given a certain amount of input energy to increase its temperature up to its saturation temperature. Beyond the saturation temperature evaporation of water takes place. Since the dissolved solid particles do not exert any partial pressure, they get separated when water evaporates. The salt is left behind and the water vapors are allowed to condense in another heat exchanger. The condensate water is then collected as the potable water for direct consumption.

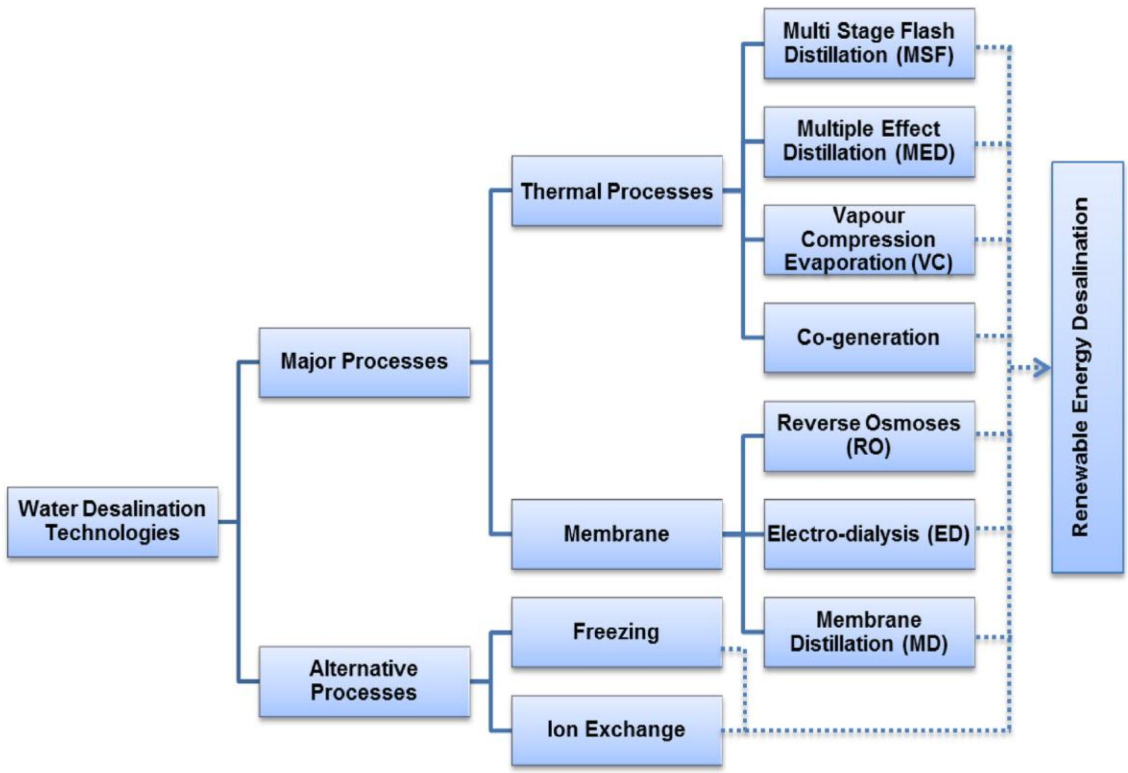

Figure 1.2 Various water desalination methods [25]

The most common thermal desalination processes are:

- Multi-stage flash distillation (MSF)
- Multiple-effect distillation (MED)
- Vapour-compression evaporation (VC)
- Solar water desalination

1.2.2 Membrane desalination

The membrane desalination processes is basically a commercial filtration process through the utilization of a permeable membrane. It is generally used for municipal water treatment such as micro-filtration and desalination. This process can be used with improved membrane for water industry, but it is mainly suitable for high return processes such as chemical separations, enzyme concentration and beverage purification. This technology uses a relatively permeable membrane to move either water or salt to induce two zones of differing concentrations to produce fresh water. A membrane is a thin film of porous material that allows water molecules to pass through it, but simultaneously prevents the passage of larger and undesirable molecules such as viruses, bacteria, metals, and salts etc. [25] Membranes are made from a wide variety of materials such as polymeric materials that include cellulose, acetate and nylon, and non-polymeric materials such as ceramics, metals and composites [25].

1.3 Solar Distillation

The World Health Organization estimates that over a billion people lack access to purified drinking water and the vast majority of these people are living in rural areas. The low population density and remote locations make it difficult to install traditional clean water solutions in those areas. The desalination processes such as reverse osmosis; multi stage evaporation and electro dialysis are energy intensive processes and are not feasible in remote areas. This difficulty can be overcome greatly by integrating the renewable energy sources with desalination process. Since solar insolation in the remote regions is generally very high so it provides us with a good opportunity to use solar energy as the energy source for water desalination. Current statistics on desalination shows that only 1% of total desalinated water is based on energy from renewable sources [25].

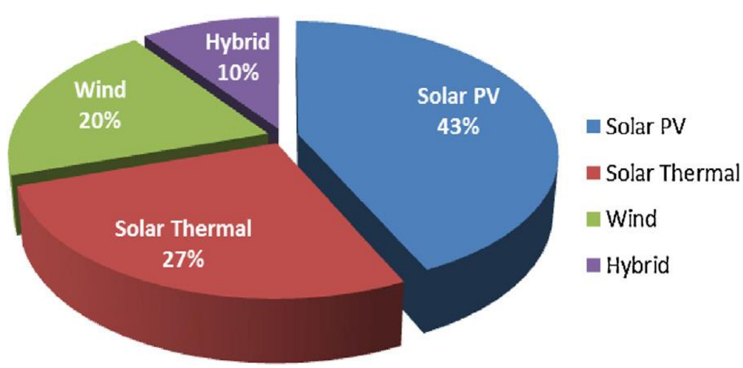

Figure 1.3 Use of renewable energy sources for water desalination [25]

Extensive research and development activities have been conducted to explore the opportunities for sustainable and feasible methods of producing drinking water using renewable energy sources. Use of solar energy in water desalination is the most promising and cost effective method to provide potable water in remote areas. This is due to the fact that solar energy is available in abundance and space requirement is not a problem in remote villages.

1.3.1 Solar still

Solar still is a device which works on the principle of solar distillation and provides potable water for direct human consumption. It is not feasible to install the commercial methods of water distillation in remote areas as availability of fossil fuel or electricity is scarce in those areas. So in order to provide clean water in remote villages, solar still has emerged as the most effective alternative. As the name suggests it is an immovable device which utilizes the thermal energy of sun light for water distillation. Solar energy is inexhaustible, available in abundance and pollution free. These features make it the best alternative among all renewable energy sources. A typical solar still works on the evaporation and condensation phenomena. Solar energy is utilized to first heat up the liquid water to evaporate it and produce vapour, then the vapour is allowed to get condensed on a tilted glass cover. The condensate is then collected as the potable water.

1.3.2 Working principle of solar still

In its simplest form, a solar still is an air tight insulated basin, containing impure water and covered with a transparent material. Basin is generally made of galvanized iron sheet (GI-sheet) and the top cover is generally made by transparent glass or plastic. Glass cover is made inclined and a collecting tray is provided at the base of inclined glass cover to collect the distillate output. The working principle of solar still is exactly same as the hydrological cycle found in nature. The solar radiations falling over the transparent glass cover gets pass through it and strike the inner surface of the basin. The inner surface of the basin is blackened so that it absorbs most of the radiation. The saline water in the basin gets heated up and begins to evaporate. Vapors start to rise leaving behind the salt and impurities. When the vapor reaches the inclined glass cover it gets condensed on the inner glass surface and the condensed water runs down due to gravity in the collecting channel. Distilled water is then taken out of the system for direct use.

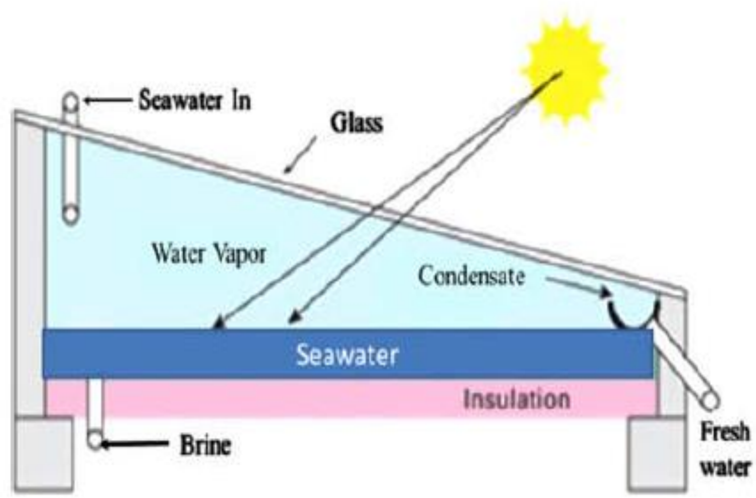

Figure 1.4 Working of solar still

Solar still is an attractive alternative for potable water production in remote areas. Main advantages of solar still are:-

- It is a low cost device and can be constructed with common materials.
- Design of solar still is very simple and it can be easily constructed by people living at remote villages.
- Its operation is pollution free and it requires very low maintenance.
- There is no requirement of skilled labour for construction or operation of the solar still.

Heat transfer in solar still is a complex process as it involves all the three modes of heat transfer. Heat flows from the sun to the solar still by radiation. Heat transfer from basin to saline water, from water vapors to glass cover and from glass cover to environment takes place by convection. Heat loss from the inside of solar still to the glass cover and side walls takes place by conduction. The only drawback of solar still is its low productivity and efficiency. Efforts have been made by various researchers towards the enhancement of productivity of the solar stills.

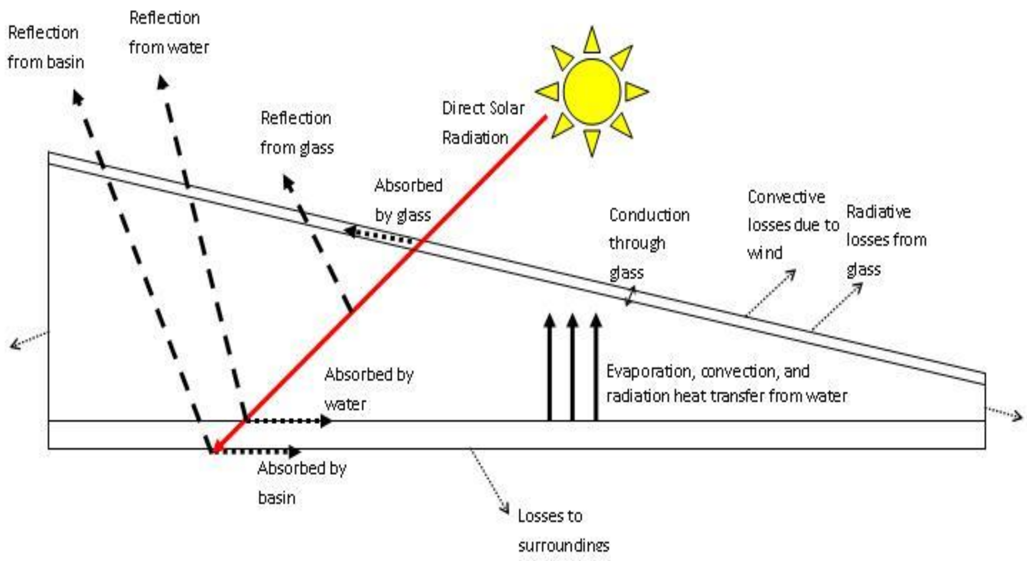

Figure 1.5 Heat transfer in solar still

1.3.3 Heat and mass transfer relations

Dunkle's relation for convective heat transfer coefficient from water to glass (h_{cw}) was employed in this work. Use of this relation is justified as the temperature within the still lies in the range of 25 to 60^0 C. Convective heat transfer coefficient is given by the following equation:-

$$h_{cw} = 0.884\left[T_w - T_g + \frac{(P_w - P_g)(T_w+273)}{268900 - P_w}\right]^{1/3} \tag{1.1}$$

Where,

$$P_w = exp\left(25.317 - \frac{5144}{T_w+273}\right) \tag{1.2}$$

And

$$P_g = exp\left(25.317 - \frac{5144}{T_g+273}\right) \tag{1.3}$$

The convective heat transfer from water to the glass cover can be expressed as:-

$$q_{cw} = h_{cw} (T_w - T_g) \tag{1.4}$$

The expressions for Evaporative heat transfer coefficient and Radiative heat transfer coefficient from water to glass are given by following equations [21]:-

$$h_{ew} = 0.016273 (h_{cw}) \left(\frac{P_w - P_g}{T_w - T_g}\right) \tag{1.5}$$

For higher operating range of temperature, the expression given by Clark [21] is generally used which is given by the following expression:-

$$h_{ew} = 0.008 (h_{cw}) \left(\frac{P_w - P_g}{T_w - T_g}\right) \tag{1.6}$$

Evaporative Heat transfer from water to glass cover is given by:-

$$q_{ew} = h_{ew}(T_w - T_g) \tag{1.7}$$

Radiative heat transfer coefficient can be determined by:-

$$h_{rw} = \varepsilon_{eff}\sigma \left[(T_w + 273)^2 + \left(T_g + 273\right)^2\right] \left[T_w + T_g + 546\right] \tag{1.8}$$

Where $$\varepsilon_{eff} = \frac{1}{\varepsilon_g} + \frac{1}{\varepsilon_w} - 1 \tag{1.9}$$

The hourly yield of a single-slope solar still can be expressed as:-

$$\dot{m}_{hourly} = \frac{q_{ew}}{h_{fg}} \times 3600 \times A_{eff} \tag{1.10}$$

However Dunkle's model is good for low temperature operations of the solar still yet there are some limitations to the Dunkle's model [19]:-

- This model was based on experimental data for low operating temperature range between 55℃ and 70℃.
- This model doesn't take into account the cavity dimensions.
- The slope of glass cover was very small (10℃), so the condensing and the evaporating surfaces were approximately parallel.
- It was originally developed for free convection of air without evaporation.

1.4 Application of CFD in Solar still

Solar still is an attractive alternative to provide drinkable water in remote areas where there is scarcity of conventional energy resources. Solar still has various advantages such as low cost, simple construction and pollution free operation. However it suffers with the challenge of low productivity of distillate water. Many researchers have studied the solar still from different angles of investigation and most of these studies have been focused towards the enhancement of productivity of solar still. Integrating the solar still with other devices like flat plate collector, solar pond or use of concentrators have been found to be effective in enhancement of still productivity but these methods greatly increases the cost of solar still. It is a big challenge to improve the productivity of still without sacrificing its inherent beauty of being a low cost device. This problem could be overcome up to a great extent by optimizing the various factors that affect the performance of solar still. This requires the study of modeling and parameter determination for an efficient design of the solar still. CFD modeling of solar still can help to a great extent in the optimization of parameters as CFD tools are capable of modeling evaporation and condensation phenomena and also provide freedom to change the parameters at will. It significantly reduces the cost and time involved in conducting a large number of experiments for the optimization of individual parameters.

1.5 Objective

In order to enhance the productivity of the solar still, a thorough study of evaporation and condensation phenomena is necessary. In this work, an attempt has been made to simulate the heat transfer phenomena through CFD software package.

The main objectives of this work are:-

- To develop and simulate a Two-phase 3-D CFD model of a single slope basin type solar still through ANSYS Workbench.
- To validate the CFD Model by comparing with the previous experimental results.
- To perform parametric analysis on the validated model.

1.6 Outline of the Report

This report has been divided into 6 chapters. A brief outline of each chapter is given in this section.

Chapter – 1 Introduction

This chapter gives a background of the problems related to drinkable water availability and the need for water purification methods. It also gives an insight about the effective methods of water purification in order to augment the increasing demand of clean water. It gives a brief description about solar distillation processes and the application of CFD in solar still. An outline of the research objectives is also given in this chapter.

Chapter – 2 Literature review

This chapter gives the summary of research work done in the area of solar distillation and solar still. Literature review has been divided into four major parts: - a) Theoretical studies on solar still, b) Experimental work done on solar still, c) CFD simulations on heat and mass transfer phenomena, d) CDF simulation on solar still. This chapter accomplishes with identification of specific objective and scope of the present work.

Chapter – 3 Geometric modeling and Mesh generation

This chapter starts with the brief background of Computational fluid dynamics and the methodology involved in solving the flow problems with ANSYS FLUENT. Further this chapter gives the details of problem formulation for CFD simulation, construction of solar still, creation of geometry and generation of mesh for the computational domain.

Chapter – 4 CFD Simulations

This chapter gives details about the solution strategy adopted for the simulation if the CFD model of solar still. Various types of boundary conditions and input parameters are specified in this chapter.

Chapter – 5 Results and discussion

This chapter gives the simulation results in qualitative as well as quantitative manner. A comparison of simulated and experiment results for the validation of the model has also been included in this chapter.

Chapter – 6 Conclusions and scope for future work

This chapter gives the conclusions made from the present work and scope for the future work.

CHAPTER 2
LITREATURE REVIEW

Solar water distillation is an oldest and cost – effective method to provide potable water in remote villages where the electricity is rather scarce. The device that carries out the solar distillation is generally known as the solar still. Solar still have many advantages such as low cost, simple construction, low maintenance etc. but it has a major disadvantage of lower efficiency and low productivity of potable water. Various experimental and theoretical works have been done by on the solar stills in order to improve the performance. In this section a review of research papers on solar stills and CFD is presented. Literature review has been divided in the following sub sections:-

- Theoretical studies on solar stills
- Experimental work done on solar stills
- CFD simulation studies in the area of heat transfer
- CFD analysis of solar still

2.1 Theoretical studies on solar stills

Sampathkumar et al. [1] in their research work presented a detailed review of active solar distillation. A comparative study was conducted on various designs of solar still and numbers of modification were made for the performance improvement of the still. In addition, a thermal was also developed for the analysis of solar still. Based on the review of various research works on solar stills over the years, following conclusions were made:-

- Active mode of operation of a solar still provides significant amount of hourly yield and is commercially viable.
- Solar still coupled with flat plate collector with forced circulation mode gives higher yield than that of the thermo-syphon mode.

- The direct steam generation parabolic through is a promising technology for solar assisted seawater desalination.
- The length of solar still, depth of water in basin, inlet water temperature and solar radiation are the major parameters which affects the performance of the still.
- The purity of the desalinated water in the tubular solar still is greater than that of a conventional one.
- In case of active solar stills, the night time productivity can be improved by using the thermal storage materials.
- Solar still with multistage basin is more effective than the still with simple basin.
- The thermal model of solar stills should be developed based on the assumption that $T_{gi} \neq T_{go}$.

Khalifa et al. [2] in their research work developed various performance correlations for basin type solar still. The effect of climatic, operational and design parameters on the performance of solar still were examined on the basis of various experimental and numerical studies. A comprehensive review of various research works has been presented. The most important parameters investigated were solar radiation, cover tilt angle, brine depth and use of dyes with the brine. The effect of each parameter was quantified and correlations were formed.

Murugavel et al. [3] reviewed the progress in the works done on single basin passive solar still to enhance its productivity. The effect of various parameters such as still orientation, glass cover inclination, cover plate material, basin material and condensing area on the still productivity were taken into consideration and critically reviewed. Important findings are summarized below:-

- Orientation of the glass cover depends upon the latitude of the place. For lower latitude places double slope solar stills are preferred over single slope solar still.
- Productivity of the solar still depends on the depth of water in the basin.
- For higher sun radiation intensity places, deep basin still is preferable, on the other hand for lower sun radiation intensity places shallow basin still is preferable.

- Black dye is suitable in deep basin solar stills in order to increase the absorption and surface heating effect.
- Rubber is very good basin material to enhance absorption, storage and evaporation effects.

Shankar and Kumar [4] presented a review article on the solar distillation process. In that extensive review of solar distillation history of solar distillation, working of solar still, advantages and disadvantages of solar still and parametric study on the solar still was presented. Moreover, various types of designs of solar still were also reviewed. Many recommendations and conclusions were drawn for solar stills based on the literature review. It was concluded that the single slope solar still is more effective than a double slope solar still in high altitude areas. Several parameters which affect the distillation rate of solar still were also investigated. Solar still was recommended as the best suited alternative to provide the clean drinkable water for remote villages.

2.2 Experimental work done on solar stills

There have been several attempts to improve the design of solar stills and to develop the mathematical model for various types of solar stills. In this section the research papers on experimental work on solar still are discussed.

Ayoub et. al [5] introduced a new and sustainable development modification in conventional solar still. A slowly rotating drum was introduced within the still cavity which allows the formation of thin water films and enhances the evaporation rate. Influence of various parameters such as the drum speed, brine depth, solar intensity, and cover cooling etc. on the still productivity was also investigated in this work. It was concluded that slow drum speed improve the daily yield of solar still.

Badran [6] studied the performance of a single slope solar still using different parameters in order to improve its productivity. Solar still productivity was increased by 51% when asphalt basin liner and sprinkler were introduced as the enhancement parameters to the conventional still. Moreover the effect of water depth on the daily yield was studied. It was found that the daily yield decreases with increase in the water depth in the basin. It was also concluded that the ambient conditions such as wind velocity and ambient temperature have a direct effect on the productivity of solar still.

Kumar et al. [7] integrated a single slope solar still with an evacuated tube collector (ETC) and operated in forced mode. In order to evaluate the performance of integrated system, a thermal model was also developed under the climatic conditions of New Delhi, INDIA. The water temperature as well as the distillate yield was increased by the integration of the ECT. The daily yield obtained was 3.47 kg/m^2 for 0.01 m basin water depth at 0.006 kg/s mass flow rate. The maximum daily energy and exergy efficiencies at optimum flow rates were found to be as 33.8% and 2.6% respectively. This work also included economic analysis of the integrated system and the cost of Rs. 2.01/kg of distilled water was estimated with a payback period of 3.7 years.

Kumar et al. [8] conducted an experimental study on various designs of solar stills. They fabricated seven different designs of solar stills that are spherical solar still, pyramid solar still, hemispherical solar still, double basin glass solar still, concentrator coupled single slope solar still, tubular solar still and tubular solar still coupled with pyramid solar still. The performance of each design was tested under same climatic conditions. From the experimental results, tubular solar still coupled pyramid solar still showed the maximum amount of productivity due to the concentrator effect.

Sahoo et al. [9] conducted an experimental study on a solar still in order to study its performance. Performance of solar still was first studied on the basis of removal of fluoride contamination from water. Solar still was found to be effective in removing fluoride contaminants as the reduction of 92–96 % was found as compared to the untreated samples. Solar still efficiency was calculated by varying the initial water depth in the basin. There was a marginal increase in still efficiency when the quantity of brackish water was increased in the basin. The efficiency of solar still was found to be increased by 4.69% and 6.05% when the basin was modified with a blackened base liner and with a blackened base liner with bottom and side thermocol insulation respectively.

Rajaseenivasan et. al. [10] conducted experimental study to compare the performance of the double and single basin type solar stills. In their research work two solar stills, single basin double slope and double basin double slope were fabricated and tested. The performance of both of the solar stills was analysed and compared a various depths of water in the basin, different wick materials, porous materials and different energy storing materials. The water production in double basin type solar still was found to be 85% more than that of single basin solar still for the same type of basin and similar operating conditions.

Murugavel et al. [11] conducted an experimental work on a double slope single basin type solar still having a basin area of 1.75 m^2 with a thin layer of 3.4 kg water in its basin. Performance of the still was compared by using the wick materials like light jute cloth, cotton cloth wick, sponge sheet and porous materials like washed natural rock. The values of cumulative water production, temperatures of glass and temperature of water were analysed for various basin wick and porous materials. It was concluded that the still with black light cotton cloth as spread material is found to be more productive. It was also concluded that the production rate is a complex function of water, glass and the difference between water and glass temperatures, basin material's volumetric heat capacity and its porosity which require further analysis.

Abdullah [12] in his research work, attempted to improve the performance of a stepped solar still by coupling it with a solar air heater. The performance of the stepped solar still integrated with solar air heater was compared with the conventional still of same basin area. In addition, the effect of water flowing over the glass cover was studied. Results showed that the water productivity was increased by 112% compared to conventional still when the stepped one was coupled with solar air heater and glass cover cooling both.

Abdallah et al. [13] conducted an experimental study introducing a number of design modifications in the conventional solar still which involves installation of reflecting mirrors on all interior sides, replacing the conventional basin with stepwise basin and coupling the solar still with a sun tacking system. Installing internal reflecting mirrors resulted in an average 30% increase in potable water production when compared to the conventional solar still. The stepped basin gave a higher water production rate with an average increment of 180% compared to a flat basin type conventional solar still. Use of sun tracking system potentially increases the water production rate. It was found that the use of tracking further improved the production rate reaching as high as 380% when compared with the classical fixed solar still.

Dev et al. [14] in their research work presented an experimental study on inverted absorber solar still along with the study of a single slope solar still at different water depth and total dissolved solid. It was observed that higher water temperatures could be achieved by using the inverted absorber solar still compared to the single slope solar still. The daily yield of potable water from inverted absorber solar still was found to be higher than that of single slope solar still at same water depth.

Taghvaei et al. [15] carried out the experiment on two solar stills at various water depths in order to analyse the long term effect of brine depth on the water production of the solar still. Two parallel solar stills were constructed and the experiments were conducted for 10 days continuously. It was found that the efficiency of the solar still increased with the decrease in water mass for first two days. After two days the overall distillate rate and the efficiency of the solar still was found to be increased by increasing the water mass in the basin. It was concluded that for the long term operations of a solar still higher water mass in the basin results in high productivity and high efficiency of the solar still.

Bhardwaj et al. [16] studied the effect of different condensing surfaces on the distillate yield of the solar still. Four types of condensing surfaces viz. Glass, PET, PC and PPM were used. The contact angle and the reflectivity of the cover were found to be the most important parameters for choosing the condensation surface of a solar still. Since glass has low contact angle in wetted region so it allows more solar radiation to pass through it consequently gives higher distillate rate than the other materials.

2.3 CFD based studies on heat transfer

Kumar and Saini [17] evaluated the performance of a solar air heater duct with artificial roughness in the form of thin circular wire in arc shaped geometry by using CFD. The effect of artificial roughness on the heat transfer coefficient and friction factor was investigated. Moreover it was concluded by CFD analysis that the combined effect of swirling motion and boundary layer separation of the fluid was responsible for the enhancement of heat transfer rate. Different turbulent models were used for CFD simulations and their results were compared. The simulation results by using K- epsilon RNG turbulent model were found to be in good agreement with standard correlations, so the model was used for predicting the heat transfer enhancement and friction factor values.

Seong-Su Jeon [18] performed CFD simulation study for bubble condensation in the sub-cooled boiling flow. The main objective of the study was to develop a CFD model for bubble condensation by using VOF model for multiphase modeling. In addition the accuracy of the VOF model was also evaluated. The CFD model for bubble condensation was developed by introducing a source term in the governing equations of VOF model by using the User Defined Functions (UDFs).

Ghorai and Nigam [19] conducted a CFD based study to analyse the two phase flow in a pipe. They used FLUENT 6.0 package for the simulations and carried out the study of gas velocity, volume fraction of liquid and interfacial roughness. Euler model was used for the multiphase modeling in CFD. Various two-phase flow profiles were studied for different velocities of gas and liquid. The simulation results on flow characteristics and shear stress distributions were compared with the experimental results in order to validate the two-phase model. Simulation results were in good agreement with the experimental results with an error of 10%. Furthermore, the correlations for the flow field characteristics in wavy stratified flow regime were also presented.

2.4 CFD analysis on solar still

Panchal and Shah [6-20] developed a multiphase, three dimension model of a solar still. They simulated the evaporation and condensation processes through ANSYS – CFX. Simulation results were compared with the actual experimental results of single basin solar still at the climate conditions of Mehsana. The simulation results of distilled water rate, water temperature, glass cover temperature, convective and evaporative heat transfer coefficient were in good agreement with the experimental results.

Rahimi et al. [21] developed a CFD based two phase three dimensional model of solar still in ANSYS-CFX. In this work volume of fluid (VOF) model was used for modeling the two phase namely liquid water and mixture of air and water vapor. No turbulence model was used as the water was taken as stagnant and vaporization rate was low. Simulation results were compared with the experimental results already available. Simulated data on temperature of water, temperature of vapor along with the heat transfer coefficient were found to be in good agreement with experimental data.

Badusha and Arjunan [22] also made a CFD model of a solar still in order to model the evaporation and condensation phenomena occurring inside the solar still. In this study the evaporation process was considered as laminar and the still was modelled at the quasi steady state condition. To model the two phase domain the VOF model framework was used and the model was simulated using ANSYS CFX. The simulation results of heat transfer coefficient, distillate output and temperatures inside the solar still were compared with the experimental results and were found in good agreement.

2.5 Conclusions drawn from literature review

The literature review on the solar still reveals that there have been several experimental studies on the solar stills in order to enhance the productivity. Theoretical studies have also been conducted in order to assess the feasibility and effectiveness of solar desalination process. There have been several attempts towards the mathematical modeling of solar still. However there have been little attempts towards the modeling of solar still through CFD tools. From the literature review following conclusions can be made:-

- Most of the research work on solar still has been focused on the enhancement of distillate yield. Various experimental works have been reported for the yield enhancement by integrating solar still with other devices such as solar air heater or the solar flat plate collector.
- Use of solar concentrators can be effective in for the enhancement of distillate yield. Other methods involve the use of wick type solar still, use of stepped basin or double basin, use of double slope solar still.

- The single slope solar still is more effective than the double slope solar still in the areas having small value of latitude angles while double slope still is more effective in high latitude areas.
- There are several factors which affect the distillation process in a solar still. Major factors are water depth in the basin, ambient wind velocity, type of solar still, solar radiation intensity, addition of dyes in the brine and initial temperature of the brine.
- Various attempts have been made towards the mathematical and numerical modeling of different types of solar stills, but very few works are focused towards the CFD modeling of solar still.

CHAPTER 3
GEOMETRIC MODELING AND MESH GENERATION

3.1 Introduction

Computational fluid dynamics, generally abbreviated as CFD, is a branch of fluid mechanics that uses various numerical methods and algorithms to solve and qualitatively analyze the fluid flow problems. In CFD analysis we utilize the computers to numerically solve various flow governing equations. In past the use of CFD was not frequent because of less computational power. Nowadays, with help of powerful computers the computational power is not a restriction anymore and the use of CFD is increasing day by day. The Application areas of CFD include aerodynamics, heat and mass transfer problems, Combustion analysis in IC engines and vibration analysis etc. Use of CFD software package reduces the efforts of conducting experimental work and it also saves a lot of time. By simulating a specific problem in CFD, we can anticipate the expected outcomes of an experiment before even conducting that experiment. This helps a great deal in research and development. With the help of a good CFD software package we can model various fluid flow regimes, Heat and mass transfer phenomena, chemical reactions, emissions from the IC engines and various other phenomena.

In CFD modeling, a virtual model of the problem domain is created which is able to obtain the experimental results. The virtual model or the problem domain is resolved in smaller cells or grids and governing equations of mass, momentum and energy conservation are applied on every grid of the problem domain. Each governing equation is then solved at each cell by CFD solver. The solution will include detailed information of the flow variables like pressure gradient, temperature gradient, flow parameters etc. at each cell.

With the advancement of computers, use of CFD modeling and simulation is getting popular day by day as it can handle a wide variety of flow problems. One other reason is the availability of high computational capacities by the use of computers. The application of CFD tools reduces the physical efforts, cost and time required to conduct the experiments.

3.2 Problem statement

From the Literature review it is concluded that a lot of experimental work has already been done on the solar stills but a little research work has been done on the CFD analysis of the solar stills. Since the CFD modeling provides the freedom to change or vary the input parameters so CFD study on solar still can be helpful to quantify the effect of various parameters affecting the performance of the solar still. This could help in enhancing the rate of water production from the solar still.

In case of solar still the CFD can be used to simulate the evaporation and condensation phenomena. It reduces the physical effort and time consumed while carrying out the actual experiment. Moreover the simulation results of solar still can be used to improve the design of solar still with a view to enhance its performance.

In this work, a two phase three-dimensional CFD model of a single slope passive solar still has been made using ANSYS Workbench. The various steps involved in creating the CFD model are discussed in the following sections of this chapter.

3.3 CFD Methodology

Computational fluid dynamics (CFD) utilizes numerical methods in order to solve and analyze fluid flow, heat and mass transfer problems. Fluid flows in various regimes are governed by partial differential equations (PDEs) which represent conservation of mass, momentum and energy. In CFD technique the governing partial differential equations are converted into a set of algebraic equations which can be solved by using digital computers to simulate the fluid flow. The governing equations of any fluid flow problem are solved with respect to some specified boundary conditions using the finite volume method as implemented in the commercial CFD codes.

CFD analysis is generally based on the solution of the approximate forms of the Navier-Stokes equation, i.e. the Reynolds-averaged NS (Navier Stokes) equations (time averaged or ensemble averaged) or the filtered NS-equations, the energy equation, the mass and concentration equations as well as the transport equations for turbulent viscosity. CFD tools are able to provide the qualitative information about the flow patterns, velocity and temperature distribution within the specified enclosed space. CFD analysis are generally performed by using the software such as ANSYS CFD, PHOENIX etc., which work on finite volume method (FVM) based code for fluid flow simulations.

CFD provides a number of advantages which contribute to the growing application of general-purpose CFD codes. These advantages include its ability to study systems where it is not possible to conduct controlled experiments and to model complex physical interactions which occur in a flow field. It can also provide comprehensive flow visualization along with a wide range of comprehensive data for qualitative as well as quantitative analysis of the flow problems.

Although the use of CFD may have some drawbacks which occur generally because of user's inexperience and are therefore not fundamental. These drawbacks are far overweighed by its benefits. So the expertise in the use of CFD codes is essential for getting correct simulation results. In addition, there may be situations where complete fundamental knowledge of all the underlying physics may not exist. In those conditions, user has to consider some inherent assumptions in the mathematical model adopted which may give rise to possible inaccuracy. Because of these inherent errors, the simulation results are generally validated through experimental results.

CFD analysis of any problem begins with the creation of a virtual model of problem domain. After creation of the geometric model the problem domain is meshed in order to discretize it. Meshing is then followed by specifying the input data, defining material properties defining the boundary conditions. After that the solution setup is initialized and calculations are run for the specific period of time. After completion of the calculations, post processing is done to take the data related to the variables of interest such as air velocity, temperatures inside the domain and other flow properties. Flow diagram on the next page represents the CFD methodology in descriptive manner.

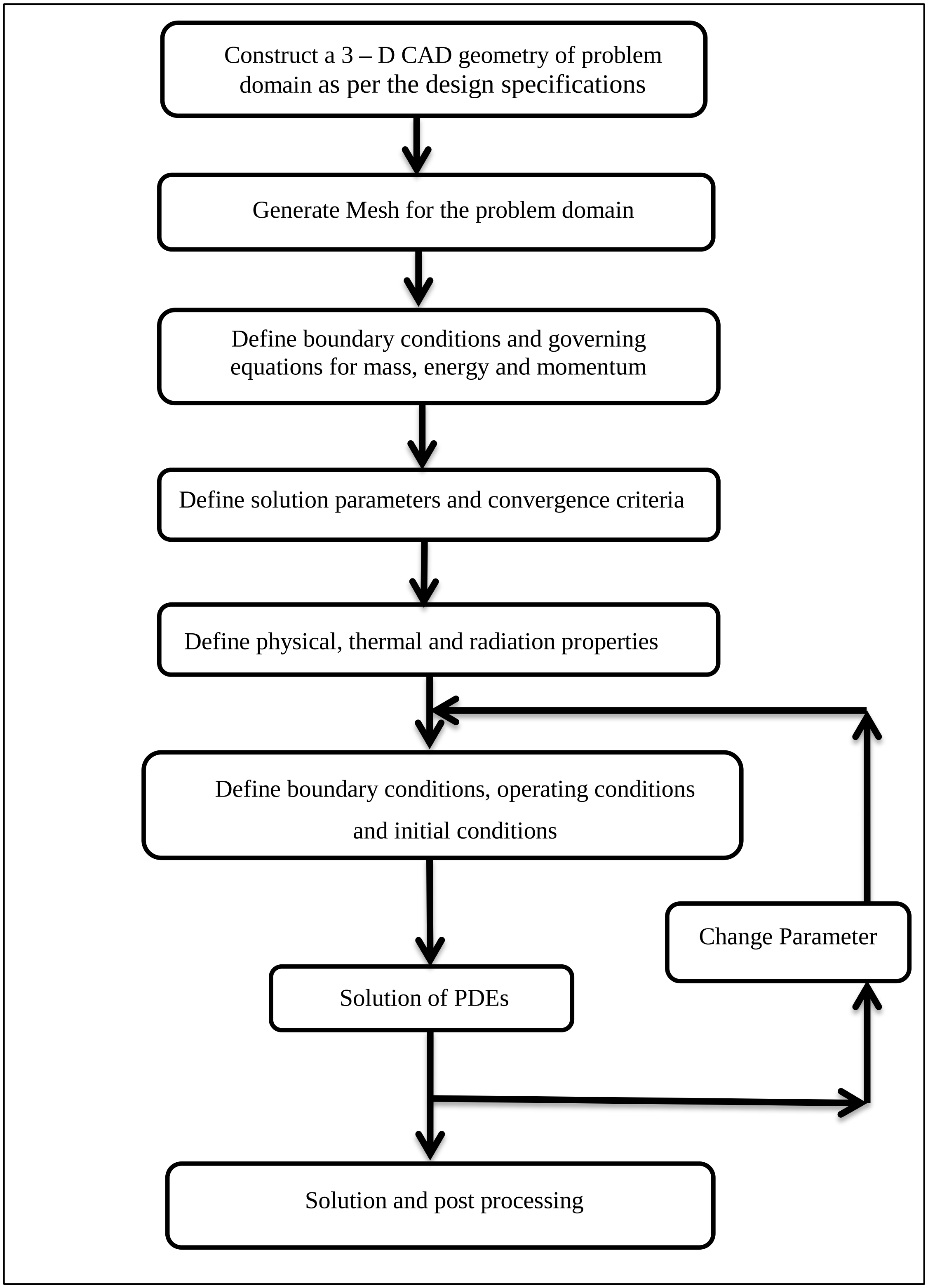

Figure 3.1 CFD Methodology

3.4 Geometric modeling of solar still

The first step in CFD analysis of any problem is creation of the geometric model of the problem domain as per the design specifications. ANSYS Workbench provides design modeler as a design tool to develop the geometric models of physical problem domain.

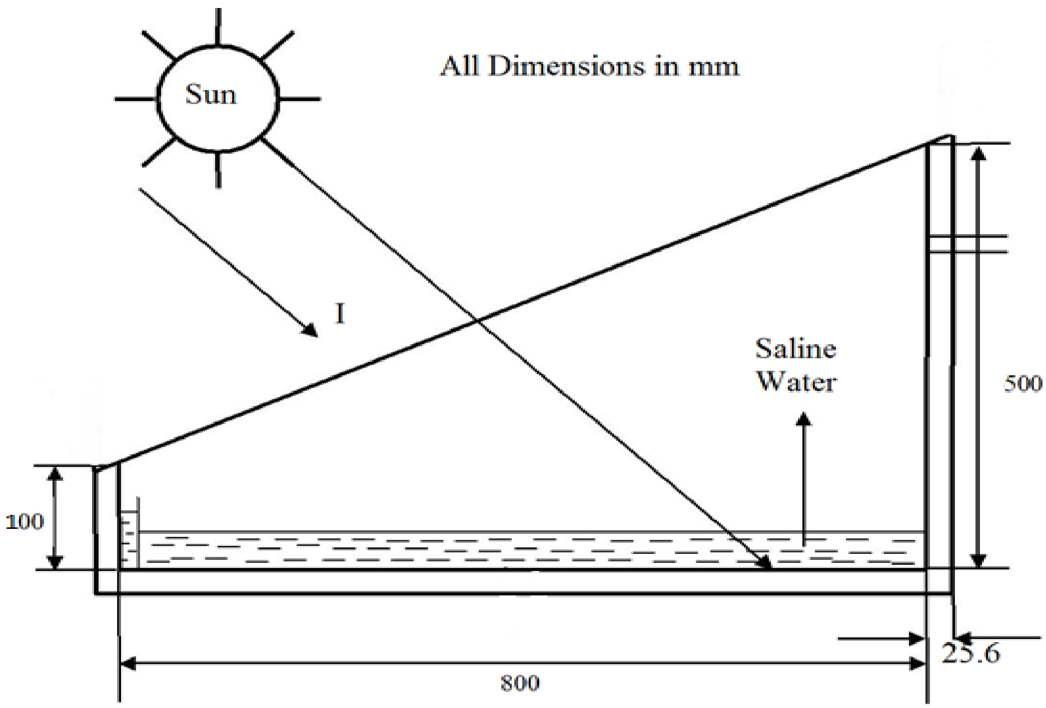

Figure 3.2 Physical dimensions of the solar still

Solar still is basically an insulated metallic box covered by an inclined transparent glass. A schematic diagram of the single slope passive solar still is shown in the figure 3.2. Solar still contains a shallow basin which is made up of GI sheet. The basin area is 0.8 m ×1.0 m. Still is constructed as a double walled body and a thick thermocol sheet is sandwiched between the GI sheet walls in order to insulate the basin. The inner basin area of solar still is painted black in order to increase its absorptivity. Mirrors are placed on the inner side walls of the basin to reflect the solar radiation towards the bottom surface of the basin. A 5 mm thick transparent glass cover is placed over the solar still. The glass cove is inclined at an angle of 26^0 towards south which is the latitude angle of Jaipur.

Design modeler provides various commands to create both 2-D and 3-D shapes. The geometric model of Solar still was created with the help of design modeler. Fig 3.3 shows the geometric model of the solar still.

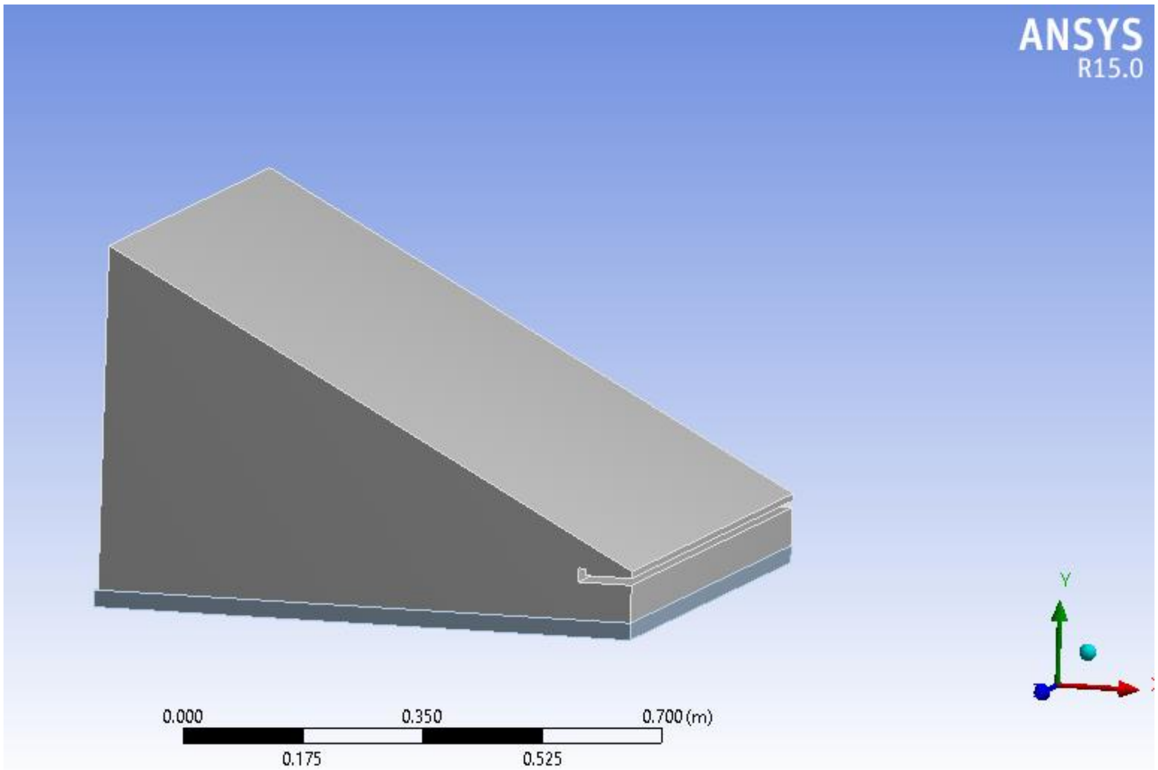

Figure 3.3 Geometric Model of the solar still

3.5 Meshing of the domain

After creation of the geometric model the next step in CFD analysis is to generate the meshing of computational domain. In mesh generation the problem domain is divided into large number of tiny cells. Various equations are solved by the CFD software for on each cell to simulate the physical phenomena. Number of cells in the domain has a great effect on the simulation results. The number of cells in the domain should be sufficiently large enough to capture the physical phenomena through simulation. At the same time, increase in the number of cells also contributes to the time required by the solver for the solution of the problem. So there is always a need to find out the optimum number of cells which can provide sufficiently accurate results along with taking moderate time for the simulation. In general, the optimum number of cells for a domain depends upon the complexity of the problem and the time available for the simulations.

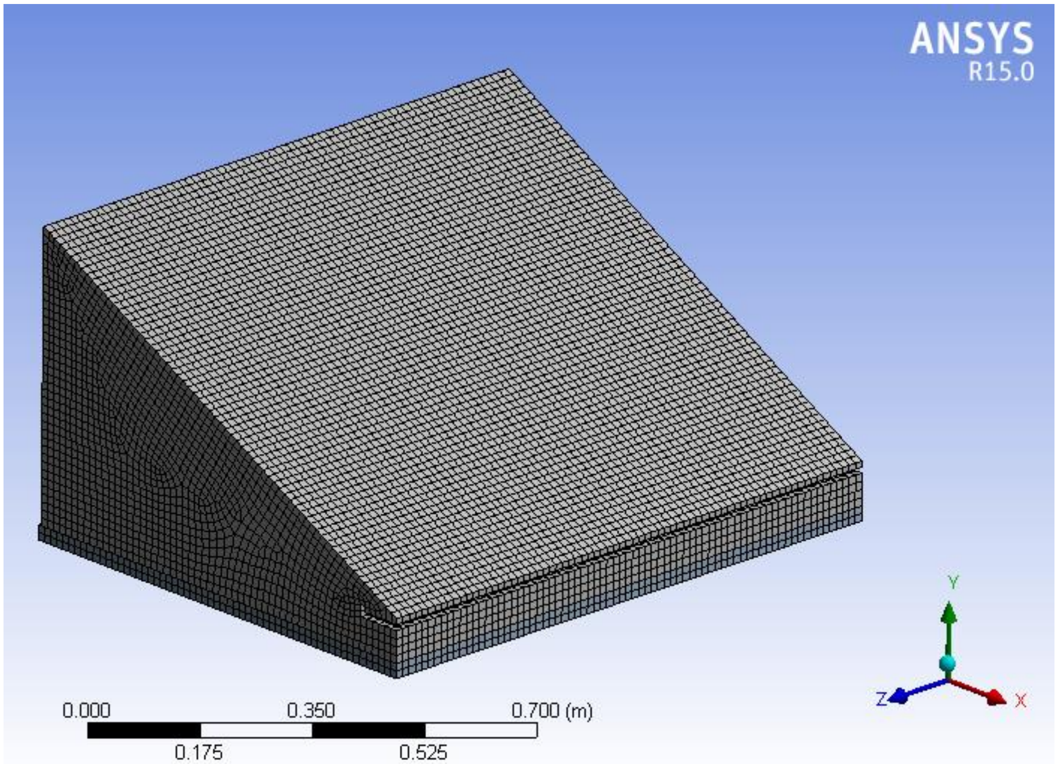

Figure 3.4 Meshing of the computational domain

A good quality mesh is essential for the accurate CFD analysis. In this study, hexahedral mesh was selected for discretization based on literature review. Meshing was done by using hexahedral elements. Hexahedral type of meshing was chosen because geometry of solar still has almost all rectangular surfaces. Since the geometry of solar still does not involve any type of curved surfaces so the hexahedral type of meshing is most suited and can provide accurate results with moderate computation time required. Figure 3.4 and fig 3.5 show the meshing of the computational domain in 3-D view. Total number of elements in the meshed domain is 100229 which are enough from the point of view of the complexity of problem in hand.

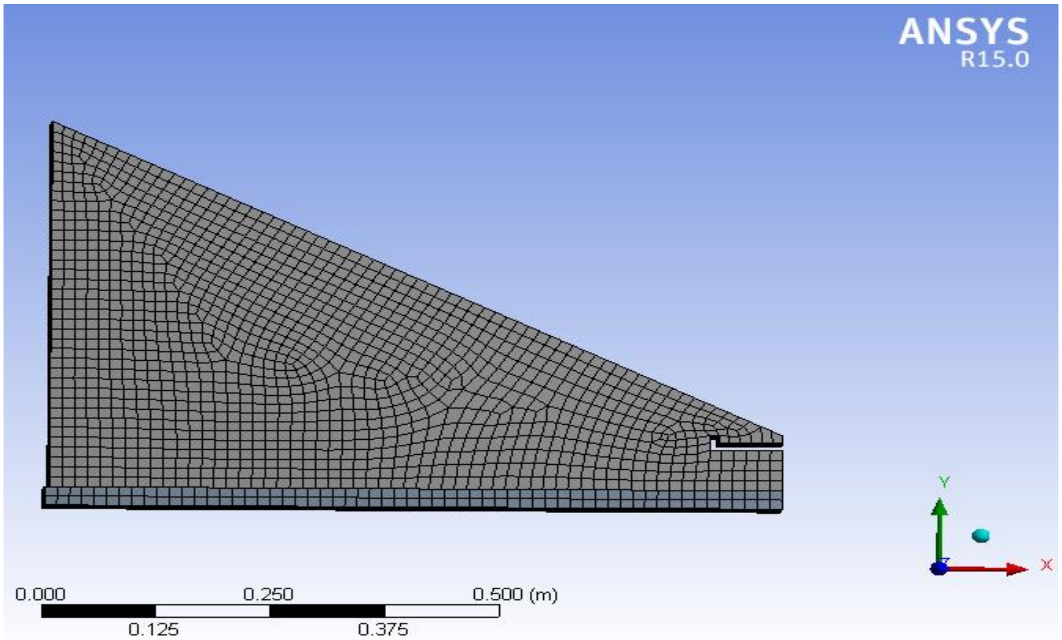

Figure 3.5 Sectional view of the meshed domain

After generating the mesh, quality of the mesh needs to be checked. It is important to check the quality of the mesh because it can affect the accuracy of the solution to a great extent. There are several parameters available with ANSYS Workbench for checking the quality of the mesh. Some of the important parameters are element quality, Skewness, aspect ratio and orthogonal quality etc. In this study the two parameters namely skewness and aspect ratio of the generated mesh have been checked.

As per the skewness criteria an element having skewness value zero is considered as the perfect element while the elements which have skewness value greater than zero are not considered to be the good quality elements. An element having skewness value 1 is in general, considered to be unviable element. In a good meshed domain there has to be very less or negligible number of elements having the skewness value equal to 1. The average value of skewness should always be less than 0.3 for a good quality mesh. As the below graph of skewness shows, most of the elements are having skewness less than 0.3. Only few and apparently negligible number of elements have skewness more than 0.3. This shows that the most of the elements in mesh of the domain has good quality. Similar to the skewness criteria aspect ratio of the elements also represents the quality of an element.

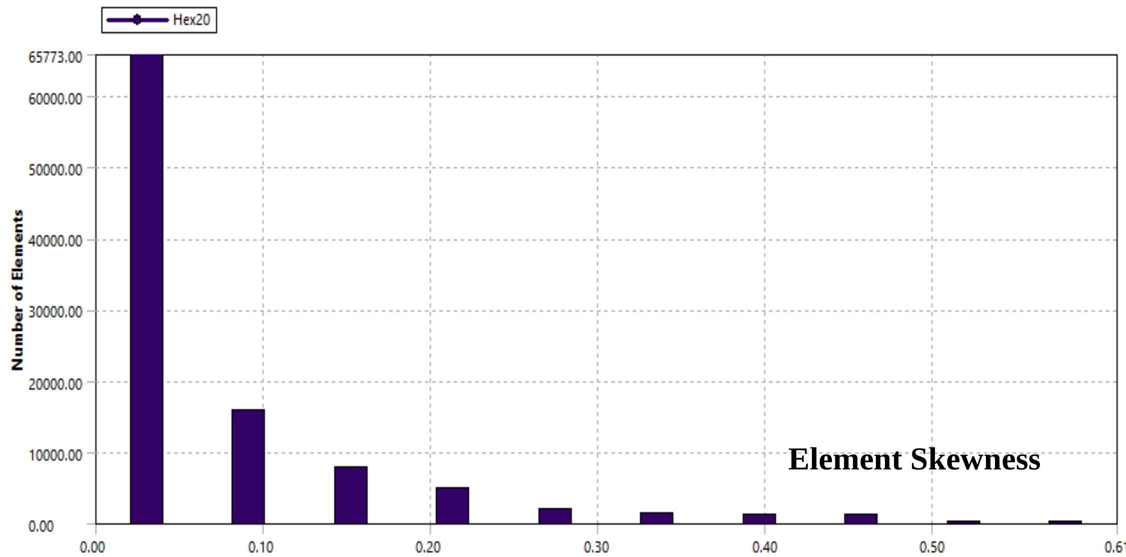

Figure 3.6 Skewness of the elements

Similarly Aspect ratio is also a parameter for the assessment of the quality of a generated mesh. The average value of aspect ratio for a good quality mesh should be less than 2. The graph of aspect ratio shows that most of the elements are having a value of aspect ratio less than 2. This indicates that generated mesh which has mostly hexahedron elements is a good quality mesh from the point of view of skewness as well as aspect ratio.

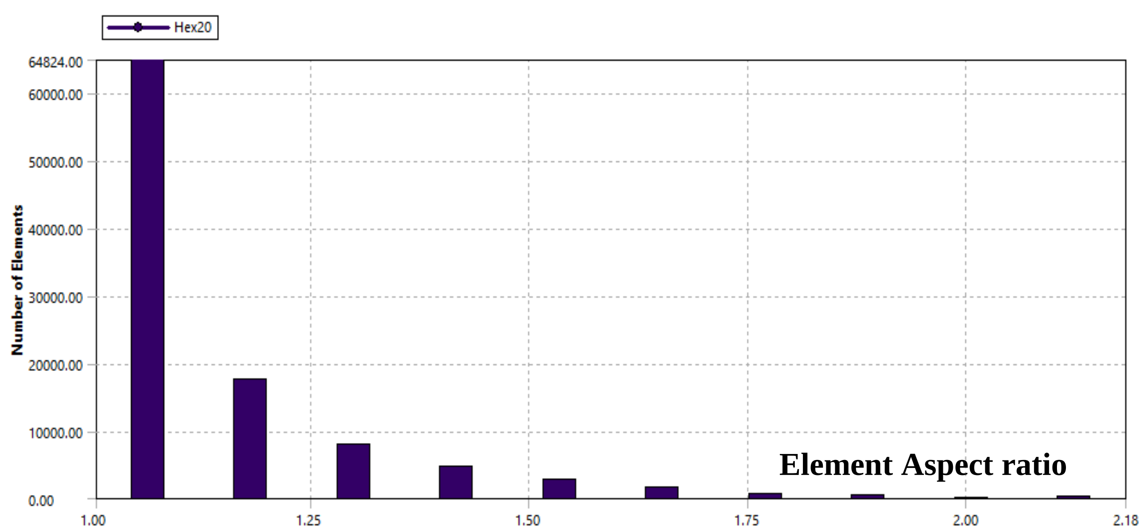

Figure 3.7 Aspect ratio of the elements

CHAPTER 4
CFD SIMULATION

Application of CFD tools for the simulation of various fluid flow problems, is gaining popularity day by day as the computational capacities have been improved with the use of digital computers. The use of CFD in engineering plays a vital role because it makes flow conditions visual within the domain. With the help of CFD tools it is easy to analyse various problems which includes turbulent fluid flows.

ANSYS FLUENT allows heat transfer within the fluid and/or solid regions in the model. Various types of complex problems involving convective heat transfer within a fluid or heat transfer in a composite system can be simulated with the help of ANSYS FLUENT. In this work ANSYS FLUENT software is used for simulations. Efforts have been made towards the CFD modeling of single slope passive solar still. In the CFD analysis of solar still, there are certain assumptions that have been taken under consideration during the simulation.

4.1 Numerical approach used

There are following four numerical approaches available for CFD simulations:-

- Finite element method
- Finite volume method
- Finite difference method
- Spectral Galerkin method

All the above methods discretize the Navier-Stokes equations for the simulation of fluid flows. Among all the four methods, Finite difference and finite volume methods are generally used by the commercial CFD packages. The CFD tool FLUENT which has been used in this work, applies the finite volume method approach for the simulation of fluid problems.

Table 4.1 various numerical approaches for CFD simulations

Approach	**Finite Element Method**	**Finite volume method**	**Finite difference method**
Discretization	Finite elements (Small domains)	Control Volumes	Elements
Based on	Piecewise representation of the solution in terms of specified basis function	Integral form of Partial differential equations	Differential form of PDE
Solutions	Continuous	Discrete	Discrete
Medium	Solid and fluid	Fluid	Fluid

4.2 Assumptions made in this work

Following are the assumptions taken in CFD simulation of solar still:-

- As the physical walls of the still are insulated so the side walls were taken to be adiabatic during simulation. In addition there is no leakage in the system.
- Velocity of air at inlet of solar still was considered to be negligible.
- As the variation in temperature is not very large, the physical properties of solids and liquids such as specific heat, thermal conductivity and density were taken as constant.
- Only filmwise condensation phenomenon occurs in place of dropwise condensation.
- As the ambient wind velocity was low, the effect of wind velocity is neglected and only free convection was taken into account.
- There is no temperature gradient across the basin water and glass cover of the solar still.

In order to include heat transfer in the model it is necessary to activate the physical models, provide input material properties and define the thermal boundary conditions that govern the heat transfer.

4.3 Boundary conditions

Defining proper boundary types and boundary conditions is essential for the accurate solution for a fluid flow problem. Most of the boundary conditions are determined by the physical phenomena but some are set by the simulation software. In this section, the type of boundary condition for each of the physical boundaries of the solar still domain is explained. Table 4.2 shows the boundary types and boundary conditions for various parts of the geometric model.

Table 4.2 Boundary types and boundary conditions

Zone name	Zone Type	Description	Heat Generation Rate (W/m^3)	wall Thickness(m)
front_wall	Wall	Adiabatic wall (Heat flux = 0)	-	0.025
back_wall	Wall	Adiabatic wall	-	0.025
side_wall_1	Wall	Adiabatic wall	-	0.025
side_wall_2	Wall	Adiabatic wall	-	0.025
glass	Wall	Convection losses	-	0.005
bottom_wall	Wall	Adiabatic wall	-	0.025

Selection of boundary conditions is an important step in the CFD simulation. Any CFD tool solves the various equations involved in the modeling on the basis of constraints putted by the boundary conditions. The real or physical boundary conditions are idealized and simplified in order to put them in the simulation. For instance, in this work the side walls of solar still which are physically insulated were considered to have the adiabatic wall boundary condition in the simulation setup.

The boundary types and boundary conditions of outlet in solar still for water were given as the pressure-outlet condition. The outlet type pressure-outlet was specified in the model. It is the appropriate outlet boundary condition for multiphase flows.

4.4 Selection of Turbulence model

Turbulence flows are characterized by chaotic motion of fluid particles. It is generally characterized in terms of irregularity, diffusivity, large Reynolds numbers, three-dimensional vorticity fluctuations, dissipation and continuum. Turbulence models are needed to solve unknown variables. The correct analysis of the problems involving turbulence 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 for any problem. Important considerations in this regard are physics involved in the flow problem, desired level of accuracy and the availability of the computational resources [26].

FLUENT provides a number of turbulence models in order to simulate the turbulent flows. Most commonly used turbulence models are:-

- Spalart-Allmaras Model
- k-ε Models
    - Standard k-ε Model
    - Renormalization group (RNG) k-ε Model
    - Realizable k-ε Model
- k-ω Models
    - Standard k-ω Model
    - Shear stress transport(SST) k-ω model
- Transition k-kl-ω Model
- Transition SST Model

The Spalart-Allmaras model is a one-equation model that solves a modelled transport equation for the kinematic eddy (turbulent) viscosity. The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. It is also gaining popularity in turbo machinery applications. In its original form, the Spalart-Allmaras model is effectively a low-Reynolds number model, requiring the viscosity-affected region of the boundary layer to be properly resolved. The Spalart-Allmaras model was developed for aerodynamic flows. It is not calibrated for general industrial flows, and does produce relatively larger errors for some free shear flows, especially plane and round jet flows. In addition, it cannot be relied on to predict the decay of homogeneous, isotropic turbulence.

Two-equation turbulence models allow the determination of both, a turbulent length and time scale by solving two separate transport equations. The standard k-ε Models in ANSYS Fluent falls within this class of models and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding [28]. The standard, RNG, and realizable k – ε models, All three models have similar forms, with transport equations for k and ε. The major differences in the models are as follows:-

- The method of calculating turbulent viscosity.
- The turbulent Prandtl numbers governing the turbulent diffusion of k and ε.
- The generation and destruction terms in the equation.

Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. The standard k-ε model is a model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The model transport equation k is derived from the exact equation, while the model transport equation for (ε) was obtained using physical reasoning. These quantities are defined as:-

$$k = \frac{1}{2}\,(u^2 + v^2 + w^2) \tag{4.1}$$

$$\varepsilon = \rho C_\mu \frac{k^2}{\mu_\tau} = \frac{k^{9/2}}{l_\tau} \tag{4.2}$$

$$\omega = \frac{\varepsilon}{k} \tag{4.3}$$

In the derivation of the k-ε model, the assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k-ε model is therefore valid only for fully turbulent flows.

As the strengths and weaknesses of the standard model have become known, modifications have been introduced to improve its performance.

The RNG k-ε model was derived using a statistical technique called renormalization group theory. It is similar in form to the standard k-ε model, but includes the following refinements:

- The RNG model has an additional term in its (ε) equation that improves the accuracy for rapidly strained flows.
- The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows.
- The RNG theory provides an analytical formula for turbulent Prandtl numbers, while the standard k- ε model uses user-specified constant values.
- While the standard model is a high-Reynolds number model, the RNG theory provides an analytically derived differential formula for effective viscosity that accounts for low-Reynolds number effects. Effective use of this feature does, however, depend on an appropriate treatment of the near-wall region.

These features make the RNG k-ε model more accurate and reliable for a wider class of flows than the standard k-ε model.

The RNG-based k-ε turbulence model is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called "renormalization group" (RNG) methods. The analytical derivation results in a model with constants different from those in the standard k-ε model and additional terms and functions in the transport equations for k and ε.

Selection of a turbulence model depends upon the complexity of problem and the desired accuracy of the result. Keeping in mind the complexity of this present problem and the available time for simulations, the k-ε model with standard wall function is selected for the simulation. The RNG k-ε model has a similar form of transport equation to the standard k-ε model:-

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_K \quad (4.4)$$

And

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k} G_k + C_{3\varepsilon} G_b - C_{2\varepsilon}\rho\frac{\epsilon^2}{k} - R_\varepsilon + S_\varepsilon \quad (4.5)$$

Where

G_k = Generation of turbulence K.E. due to mean velocity gradients

G_b = Generation of turbulence K.E. due to buoyancy

Y_M = Fluctuating dilatation in compressible turbulence to the overall dissipation rate

$C_{1\varepsilon}, C_{3\varepsilon}, C_{2\varepsilon}$ = Model constants

4.5 Selection of Multiphase Model

A large number of fluid flows encountered in practical applications involve a mixture of phases. Physical phases of matter are gas, liquid, and solid, but the concept of phase in a multiphase flow system is applied in a broader sense. In multiphase flow, a phase can be defined as an identifiable class of material that has a particular inertial response to and interaction with the flow and the potential field in which it is immersed.

Advances in computational fluid mechanics have provided the basis for further insight into the dynamics of multiphase flows. Currently there are two approaches for the numerical calculation of multiphase flows: -

- The Euler-Lagrange approach
- The Euler-Euler approach.

In the Euler-Euler approach, the different phases are treated mathematically as interpenetrating continua. Since the volume of a phase cannot be occupied by the other phases, the concept of phasic volume fraction is introduced. These volume fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of equations, which have similar structure for all phases. These equations are closed by providing constitutive relations that are obtained from empirical information, or, in the case of granular flows, by application of kinetic theory [26].

In ANSYS Fluent, three different Euler-Euler multiphase models are available:-

- The volume of fluid (VOF) model
- The mixture model
- The Eulerian model.

The VOF model is a surface-tracking technique applied to a fixed Eulerian mesh. It is designed for two or more 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. Applications of the VOF model include stratified flows, free-surface flows, filling, sloshing, the motion of large bubbles in a liquid, the motion of liquid after a dam break, the prediction of jet breakup (surface tension), and the steady or transient tracking of any liquid-gas interface.

The Eulerian model is the most complex of the multiphase models in ANSYS FLUENT. It solves a set of n momentum and continuity equations for each phase. Coupling is achieved through the pressure and interphase exchange coefficients. The manner in which this coupling is handled depends upon the type of phases involved; granular (fluid-solid) flows are handled differently than nongranular (fluid-fluid) flows. For granular flows, the properties are obtained from application of kinetic theory. Momentum exchange between the phases is also dependent upon the type of mixture being modelled. Applications of the Eulerian multiphase model include bubble columns, risers, particle suspension, and fluidized beds.

The mixture model is designed for two or more phases (fluid or particulate). As in the Eulerian model, the phases are treated as interpenetrating continua. The mixture model solves for the mixture momentum equation and prescribes relative velocities to describe the dispersed phases. Applications of the mixture model include particle-laden flows with low loading, bubbly flows, sedimentation, and cyclone separators. The mixture model can also be used without relative velocities for the dispersed phases to model homogeneous multiphase flow [26].

Mixture model is used in this study as multiphase model. It is selected because it allows modeling of evaporation and condensation phenomena in the closed domain. Also this model is not very sophisticated and requires reasonable computational time.

4.6 Selection of Radiation model

ANSYS FLUENT provides five radiation models that allow the inclusion of radiation heat transfer, with or without a participating medium, in heat transfer simulations. Heating or cooling of surfaces due to radiation and/or heat sources or sinks due to radiation within the fluid phase can be included in the model using one of the following radiation models.

- Discrete Transfer Radiation Model (DTRM)
- P-1 Radiation Model
- Rosseland Radiation Model

- Surface-to-Surface (S2S) Radiation Model
- Discrete Ordinates (DO) Radiation Model

In addition to these radiation models, ANSYS Fluent also provides a solar load model that includes the effects of solar radiation simulations. For optical thickness greater than 3, the Rosseland model is cheaper and more efficient. For high optical thickness cases, a second-order discretization scheme for the DO model is recommended.

For certain problems one radiation model may be more appropriate than the others. When deciding which radiation model to use, fluent theory guide suggests considering some thumb rules listed in the table 4.3. In this study, Rosseland model with the solar loading and solar ray tracing is used.

Table 4.3 Selection of appropriate radiation model [26]

Model	Optical thickness	Non-Grey radiation
P-1 Model	Optical thickness>>1	Allowed
Rosseland Model	Optical thickness>3	Not allowed
DO Model	Full range of optical thickness	Allowed
DTRM Model	Optical thickness<1 (optically thin)	Not allowed
S2S Model	Full range of optical thickness	Not allowed

4.7 Input parameters

There are certain values of parameters such as material properties, initial values of parameters that are needed to be inserted as input parameters to the CFD tool. These parameters may vary from problem to problem. Thermo-physical properties including density, thermal conductivity and specific heat capacity of the materials, used for CFD simulation of solar still are shown in Table 4.4 Except for GI sheet and Glass, other materials were selected from a database stored within Fluent. Properties of GI sheet and glass were defined externally.

Table 4.4: Physical and thermal properties of materials used in simulation

Material	**Density ($kg\ m^{-3}$)**	**Specific heat capacity ($J\ kg^{-1}\ K^{-1}$)**	**Thermal conductivity ($W\ m^{-1}\ K^{-1}$)**
Air	1.225	1006	0.0242
Glass	2235	750	1.15
GI Sheet	7196	502	55

The values of other input parameters are given in the table 4.5.

Table 4.5 Input parameters

Function	**Specification**		
Solver	Space	3 D	
	Time	Unsteady State, Ist-order, Implicit	
	Turbulence model	k-ε 2 equation with standard wall function	
	Multi-phase model	Mixture model	
	Radiation Model	Rosseland model with Solar loading	
Materials	Solid	GI sheet, Glass	
	Fluid	Air, water-liquid, water-vapor	
Phases	Three phase	Primary phase	Air
		Secondary phase	Water-liquid
			Water-vapor

<table>
<tr><td>Cell Zone conditions</td><td colspan="2">Source term – Energy source (UDF – Appendix-B)</td></tr>
<tr><td rowspan="3">Operating condition</td><td>Operating pressure</td><td>1.01 bar</td></tr>
<tr><td>Gravity</td><td>Y – Direction</td></tr>
<tr><td>Operating temperature</td><td>288.16 K</td></tr>
</table>

4.8 Numerical solver used

ANSYS FLUENT provides two major numerical solvers; namely, segregated solver and coupled solver (implicit and explicit type). In either of these methods, FLUENT solves the governing integral equations for the conservation of mass; momentum and energy In addition to these equations, some other equations involving scalar quantities such as, turbulence and chemical species are also solved by FLUENT whenever necessary. In case of implicit type algorithm, the individual governing equations for the variables such as velocity, temperature, pressure, turbulent kinetic energy, etc. are solved separately one after another. Each governing equation, while being solved, is "decoupled" or "segregated" from other equations. The segregated algorithm is memory-efficient, since the discretized equations need only be stored in the memory one at a time. Due to memory efficiency of the solver, implicit approach has been considered in the present case.

4.8.1 Solution technique

Solution of the equations governing equations along with the scalars (e.g. temperature, pressure, species concentrations) requires a discretization scheme. Two discretization schemes are relevant for this present work:-

- First Order Upwind, in which cell face values are set equal to the cell centre value in the cell up-stream, and
- Second Order Upwind, in which the cell face values are calculated using a Taylor Series expansion to give an increased range of influence of the surrounding cells.

The ANSYS FLUENT User's Guide advises that the First Order upwind solution scheme gives a stable solution with a good rate of residual convergence. The limitation of this scheme is that the accuracy of the solution may not be satisfactory. So first order upwind scheme can be used where high accuracy is not the prime concern. On the other hand the Second Order upwind scheme provides highly accurate simulation results. But with the use of second order upwind solution scheme time requirement for the simulation increases greatly. Therefore by taking into consideration for the calculation capacity and the time availability, first order upwind solution scheme was employed in the present work.

4.8.2 Convergence criteria

The convergence criteria are specified pre-set values for residuals which determine the convergence of the solution. To reach convergence, residuals were monitored for the X, Y and Z velocity; continuity; energy; turbulent kinetic energy (k); kinetic energy dissipation rate (ε). The convergence criterion for energy equation is 10^{-6}. Convergence criteria for all other variables are taken as 10^{-3}. Convergence criteria are set with the assumption that once the solution reaches convergence it will no longer change with more iterations.

Table 4.6 solution Parameters

<table>
<tr><th>Function</th><th colspan="3">Specification</th></tr>
<tr><td rowspan="5">Solution Methods</td><td>Pressure velocity coupling</td><td colspan="2">PRESTO</td></tr>
<tr><td rowspan="4">Spatial Discretization</td><td>Pressure</td><td>Standard</td></tr>
<tr><td>Momentum</td><td>First order Upwind</td></tr>
<tr><td>Volume Fraction</td><td>First order Upwind</td></tr>
<tr><td>Energy</td><td>First order Upwind</td></tr>
<tr><td rowspan="4">Solution Controls</td><td>Equations</td><td colspan="2">Continuity, momentum, energy, Turbulence kinetic energy, kinetic energy dissipation rate</td></tr>
<tr><td rowspan="3">Under relaxation factor</td><td>Pressure</td><td>0.4</td></tr>
<tr><td>Momentum</td><td>0.4</td></tr>
<tr><td>Volume fraction</td><td>0.5</td></tr>
</table>

After defining the parameters in Fluent, the solution was initialized. Iterations were carried out depending upon the ease of convergence and time required to get the results. Table 4.6 shows the solution parameters defined for the simulation of Solar still. Simulation results are discussed in next chapter.

CHAPTER 5

RESULTS AND DISCUSSION

CFD simulation of the single slope passive solar still has been carried out in this work. Simulation results in the form of various contours and graphs have been presented in the subsequent sections of this chapter.

Transient simulation of the solar still was carried out on 19th May from 08:00 hrs. to 18:00 hrs. Various contours of static temperatures and volume fraction of water phase have been presented. In order to validate the model, simulated results are compared with the experimental results taken on 19th May 2014 at MNIT Jaipur.

5.1 Temperature profiles

In case of solar still, temperatures attained by glass cover, water in the basin and the interior of the still play vital role for the distillation of water. In general, amount of distillate produced by solar still depends upon the temperature difference between water in the basin and glass cover. Temperature contours inside the solar still are drawn at the X-Y plane passing through Centre of the still and parallel to its side walls. Temperature profiles of interior temperature and the glass temperature are shown at different time intervals. In the temperature contours a common range from 314 K to 360 K for temperature was chosen for appropriate representation of the contours.

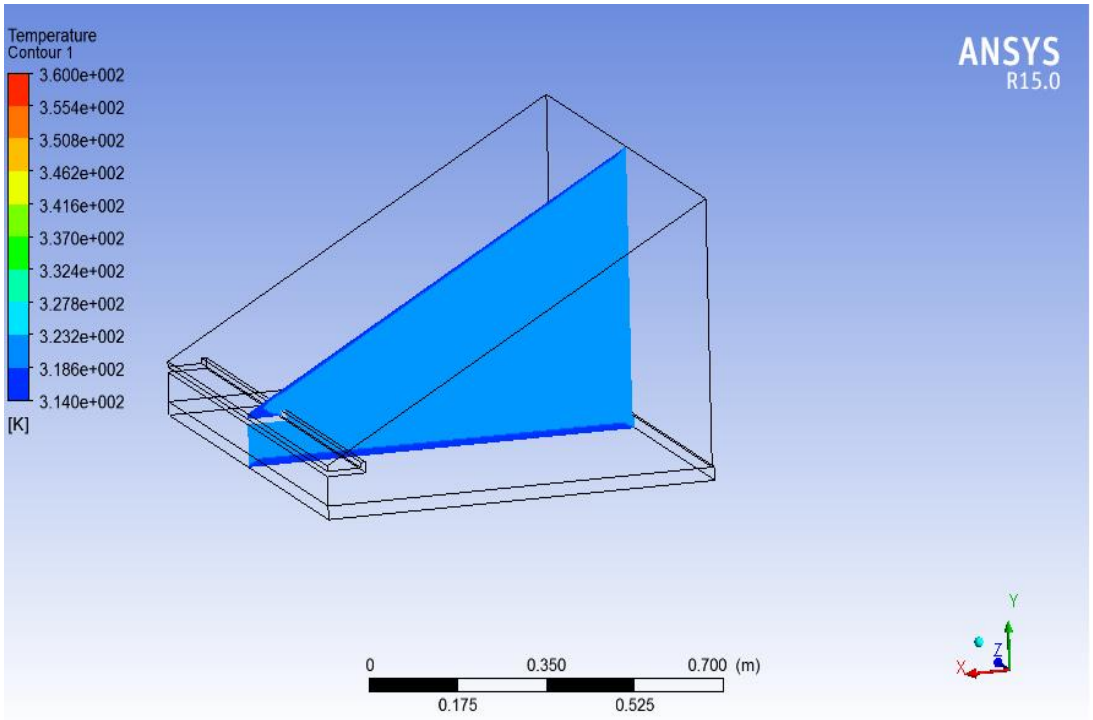

Figure 5.1 Contour of interior temperature at 09:00 hrs

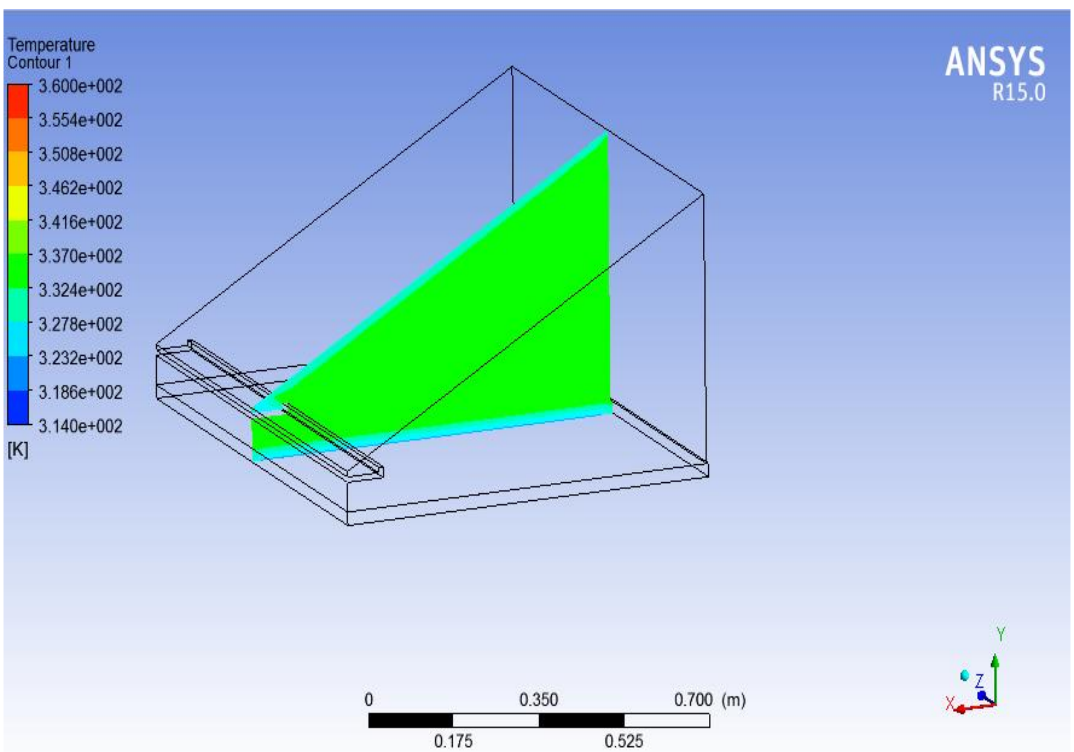

Figure 5.2 Contour of interior temperature at 10:00 hrs

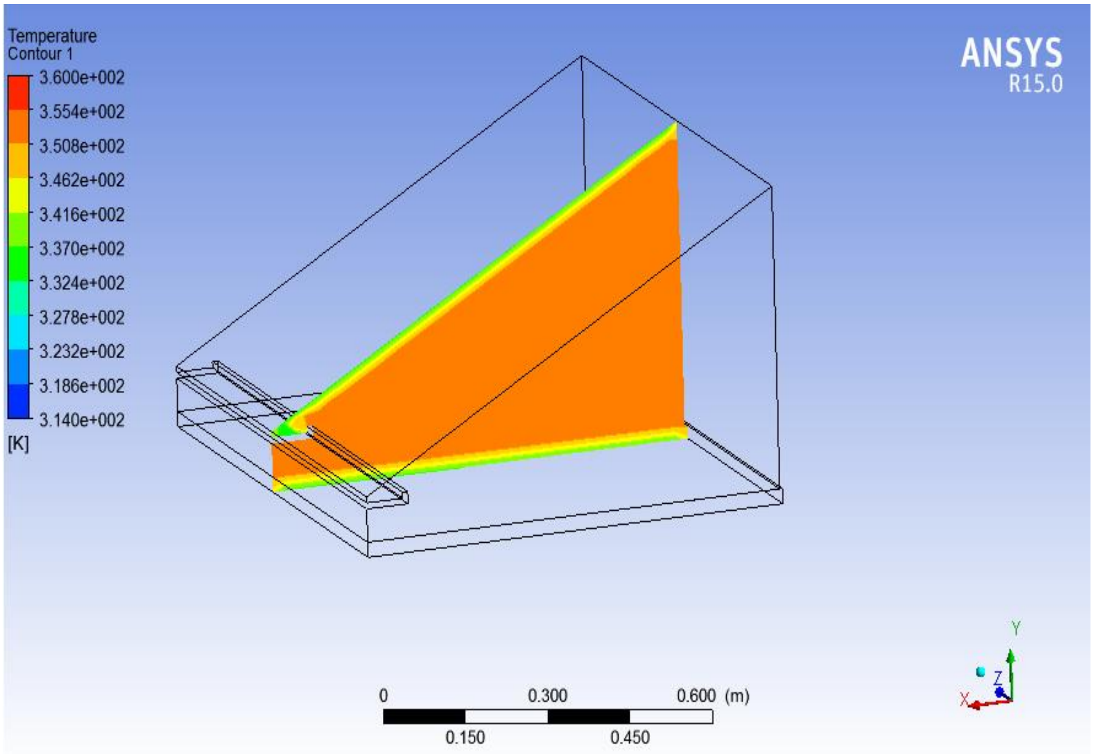

Figure 5.3 contour of Interior temperature at 12:00 hrs

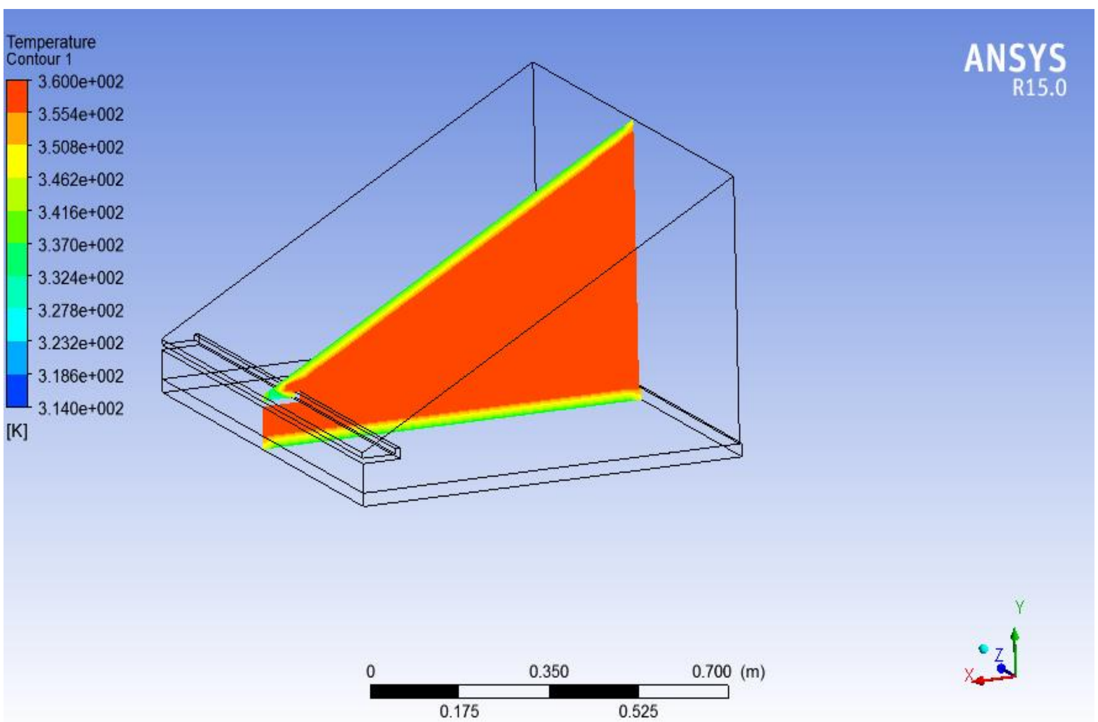

Figure 5.4 contour of Interior temperature at 14:00 hrs

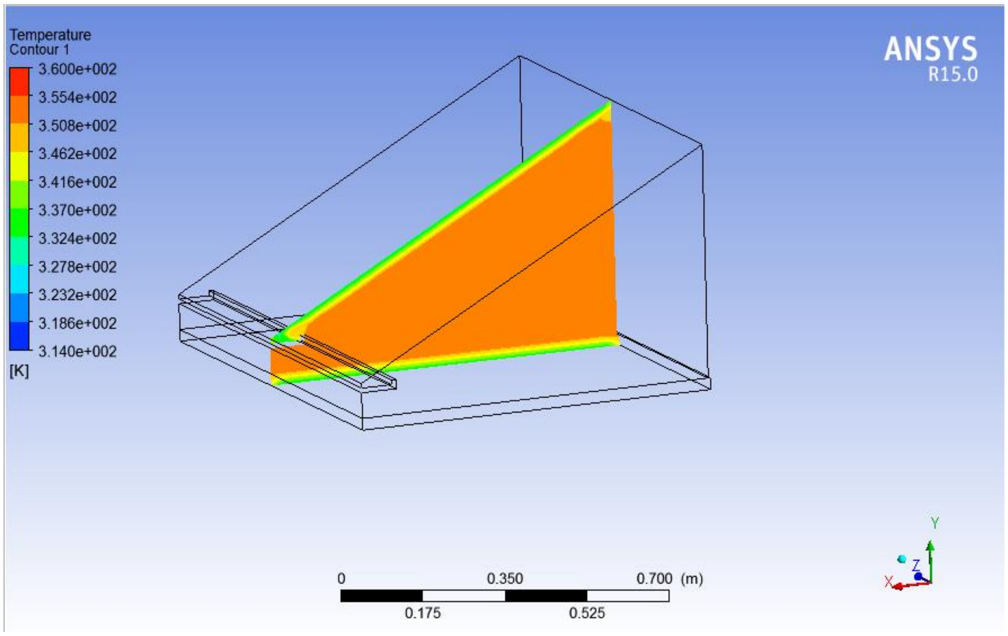

Figure 5.5 contour of Interior temperature at 16:00 hrs

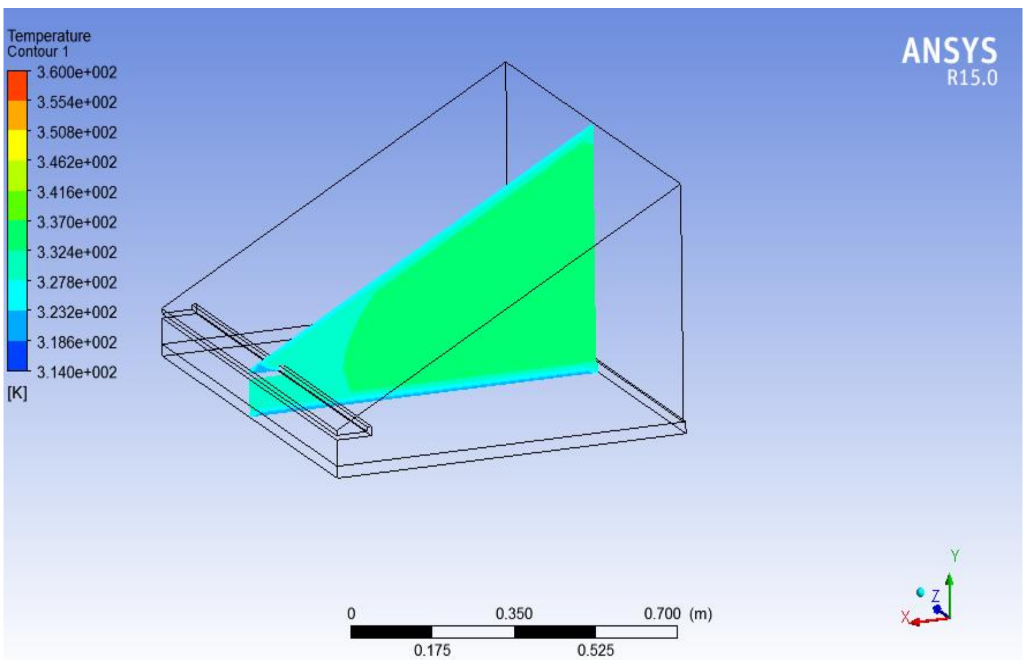

Figure 5.6 contour of Interior temperature at 18:00 hrs

The contours of interior temperature of solar still show that:-

- Within the solar still the temperature of water begins to rise as the solar radiations falls in the basin. After some time water gets heat up and evaporation takes place which results in the vapor formation in the still as well as increase in the interior temperature. Contours of interior temperature show the increment in the interior temperature of solar still with time.

- The temperature inside the solar still increases as the intensity of solar radiation increases. Interior temperature of still increases monotonically up to 13:00 hrs and after that it decreases monotonically. Basically the interior temperature of the still follows the pattern of solar radiation falling over the glass cover.
- It is clear from the contours of interior temperature that temperature of the mixture is almost uniform in the still. This is because after one hour of operation the hot vapors have acquired almost all the space in the still.

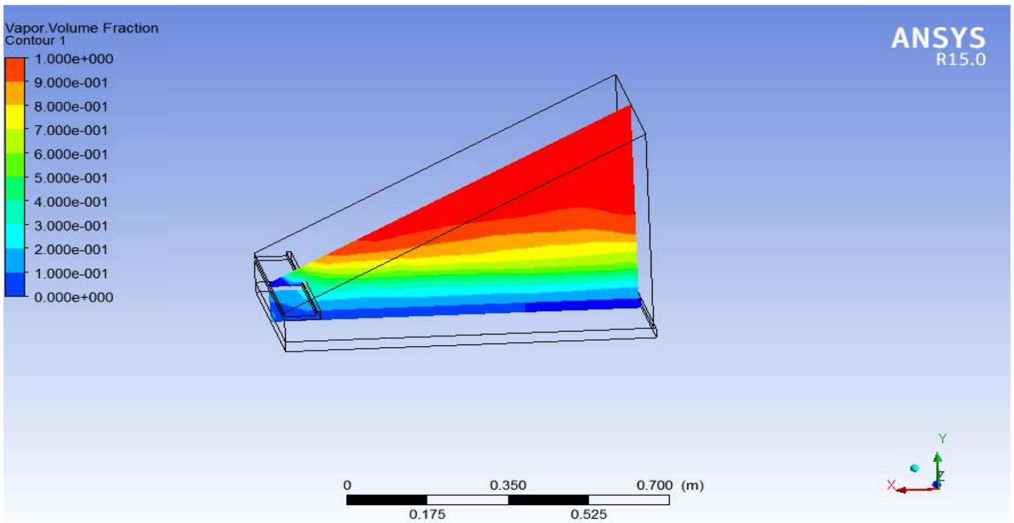

Figure 5.7 Contour of vapor volume fraction at 12:00 hrs

- The contours of glass temperature are also shown at various time intervals. Since the orientation of mesh was defined in a way that the still always faces south direction, the effect of movement of sun with time of the day, can be clearly seen on the temperature profile of the glass cover.
- As the solar intensity increases with time the temperature of glass cover also increases up to the maximum at 13 hrs after that the temperature of glass cover decreases as the solar intensity decreases after reaching a maximum value. So the

glass temperature also follows the pattern of solar intensity falling over the glass cover.

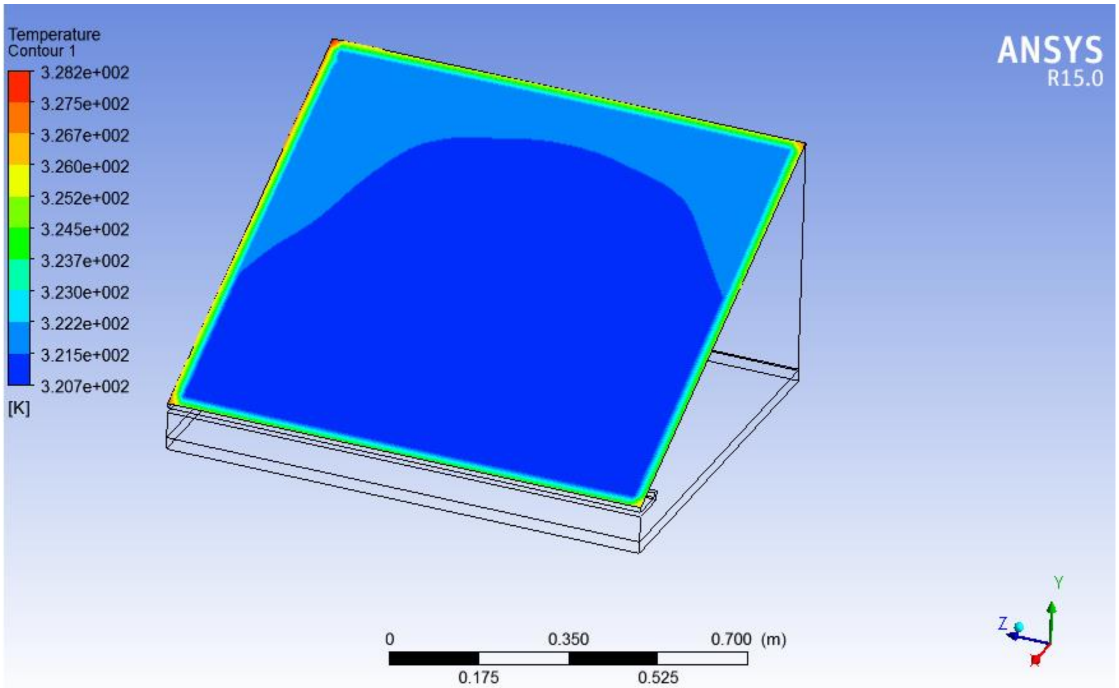

Figure 5.8 Glass temperatures at 10:00 hrs

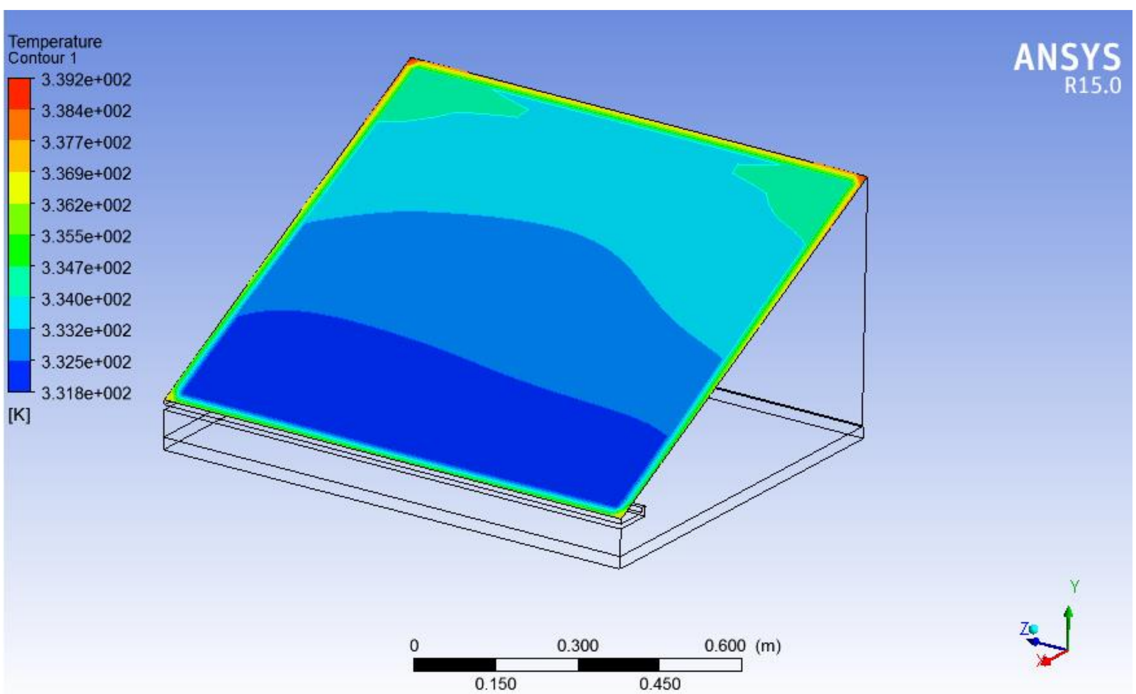

Figure 5.9 Glass temperatures at 12:00 hrs

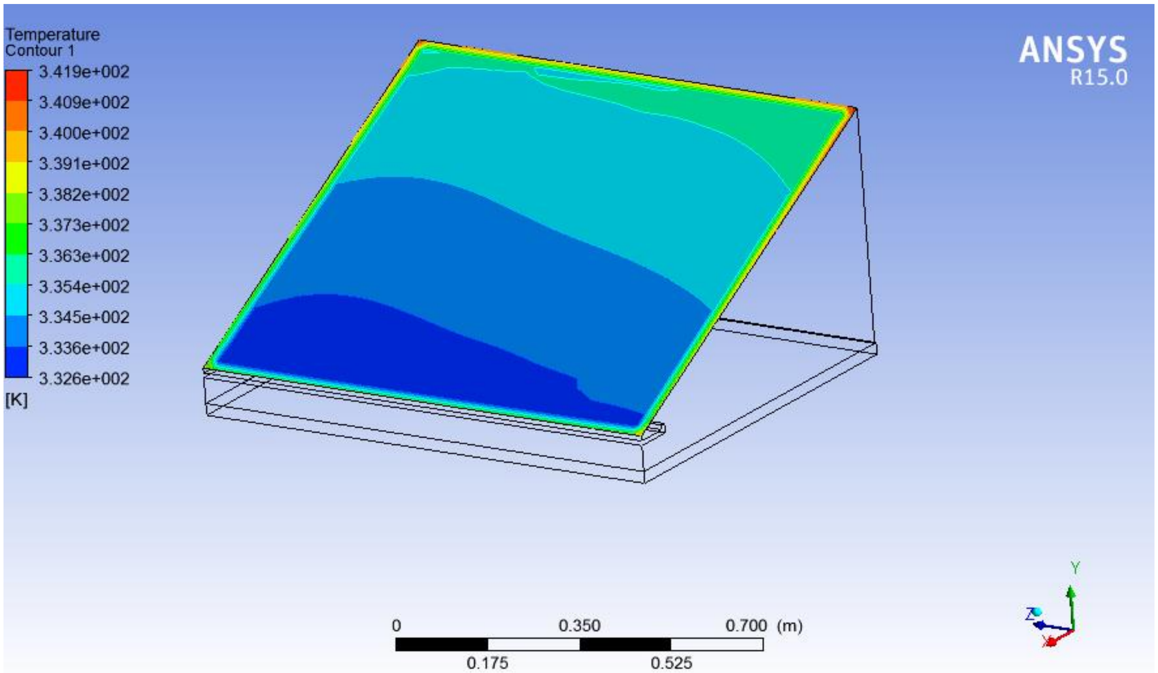

Figure 5.10 Glass temperatures at 14:00 hrs

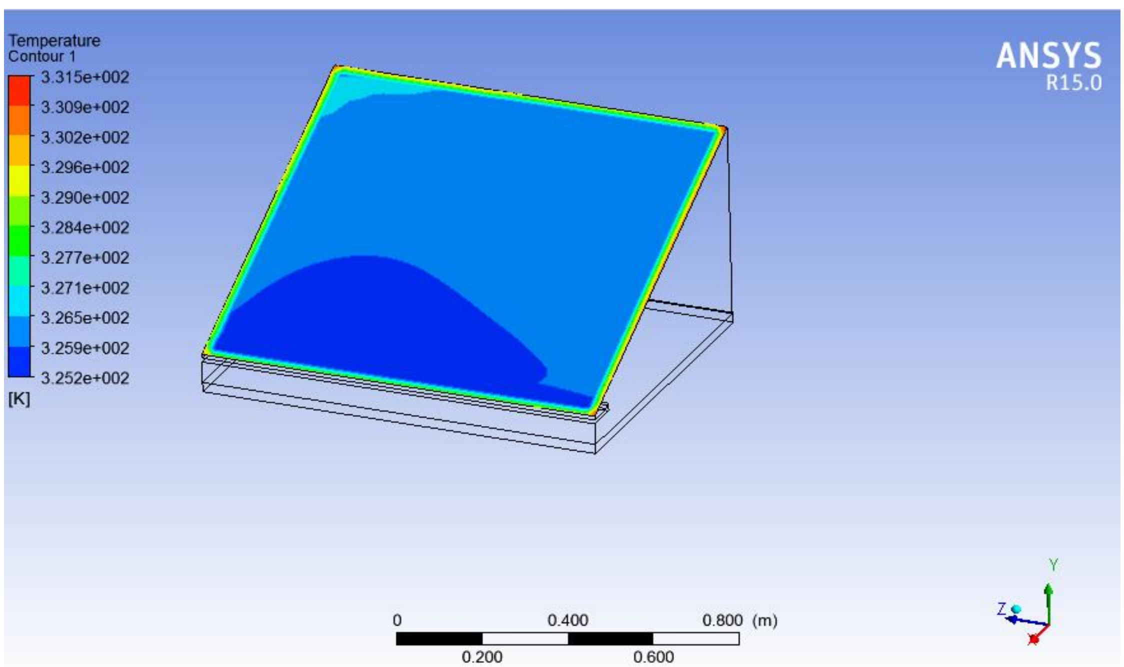

Figure 5.11 Glass temperatures at 17:00 hrs

- From the contours of glass cover temperatures it can be noticed that the temperatures on the lower end of glass cover are relatively lesser than the temperature at the upper part of the glass cover. This is due to the fact that distilled water slides down on the inside of glass cover from the upper part to the lower end of the glass cover.
- Figure 5.12 shows the contour of water volume fraction at the distillate channel. Water volume fraction in distillate channel is nearly 0.7 which indicates that the distillate water gets accumulated in the channel after sliding down from the tilted glass surface.

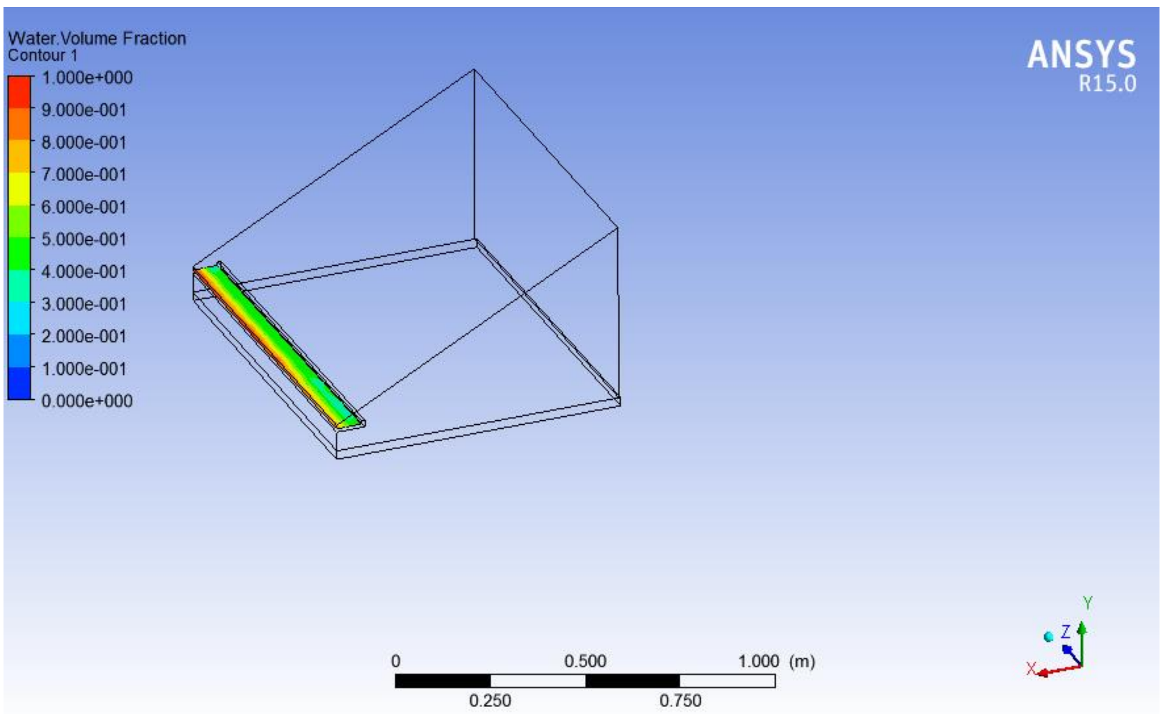

Fig 5.12 Contour of water volume fraction at the distillate channel

5.2 Validation of the model

For the validation of CFD model of solar still, the simulated values of temperatures were compared with the experimental data already available. Experimental data of the glass temperature, mixture temperature and water temperature taken on 19th May 2014 was used for the comparison with the simulated temperature data. Figure 5.13 shows the variation of glass cover temperature with time of the day. It shows that the temperature of glass cover increases from 8:00 hrs to 13:00 hrs about monotonically and after that it decreases. This trend is expected as it is analogous with the trend of solar radiations intensity.

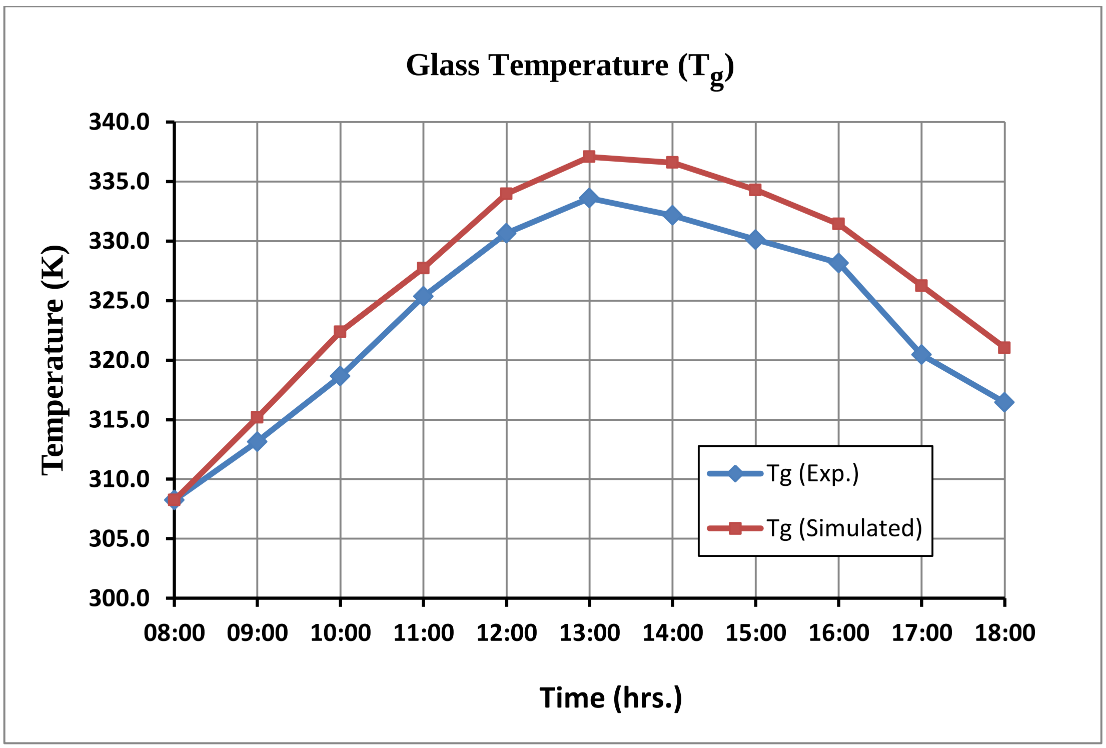

Figure 5.13 Variation of glass temperature with time

Fig 5.13 also shows the comparison between the simulated and experimental values of glass temperature. However, the experimental and simulated results both follow same pattern but there is a small difference between the two values at each step of time. This difference is attributable to the fact that CFD tool considers the ideal characteristics of glass which may differ from the actual properties of the glass used in experiment. In simulation results there is also no consideration of natural attenuation which is also a factor that induces the difference between simulated and experimental values. Figure 5.14 shows the comparison between experimental and simulated values of the temperature of water in the basin. Since the basin water takes necessary energy require for evaporation from the solar radiations, the temperature of water also follows the trend of solar radiation intensity.

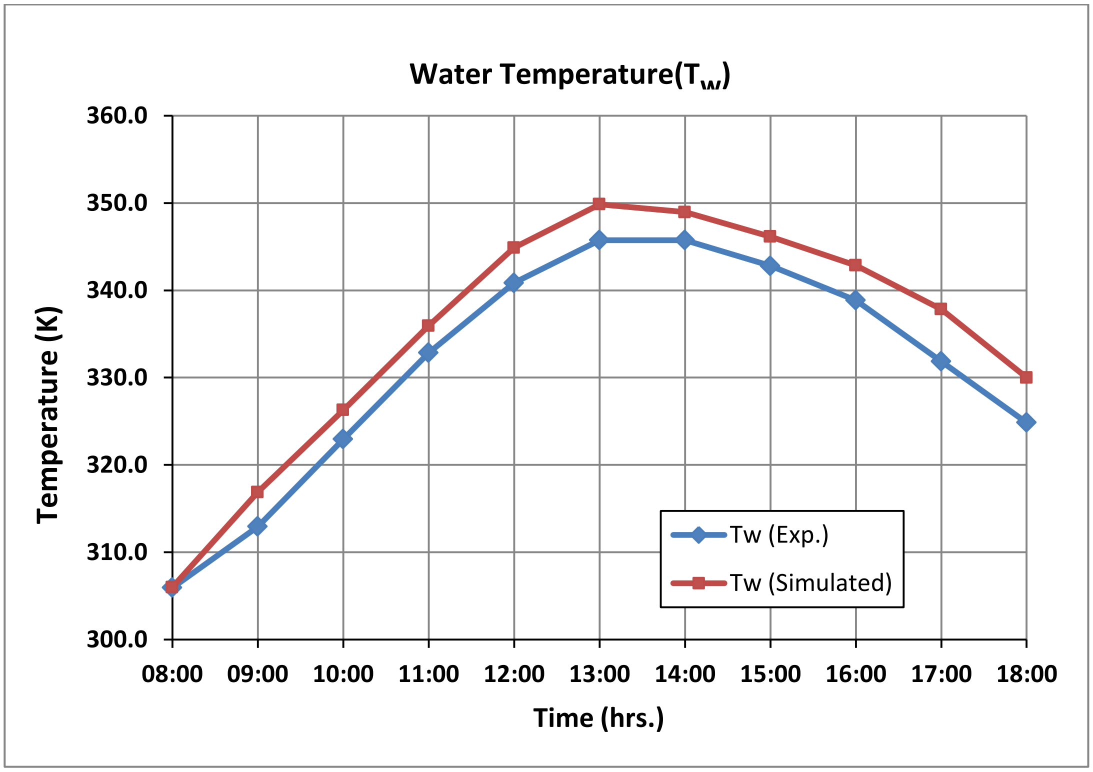

Figure 5.14 Variation of water temperature with time

Similarly the comparison of the simulated values of vapor temperature with the experimental values of vapor temperature has been shown in fig 5.15. Simulated as well as experimental data, both follow the almost the same trend. The simulation results have been compared with the experimental data taken on 19th May 2014. Although the temperature variation follows the same pattern yet there is a slight difference in the two values. The temperatures attained in the simulated results are slightly higher than the temperatures attained in the experimental data. The difference can be explained with the fact that FLUENT considers the ideal boundary condition of insulated side walls while in reality side walls are not perfectly insulated and loses significant amount of heat. In addition to that the CFD tool does not take into account for the natural attenuation. Example of natural attenuation could be the collection of dust particles on the glass cover, varying velocity of ambient air etc.

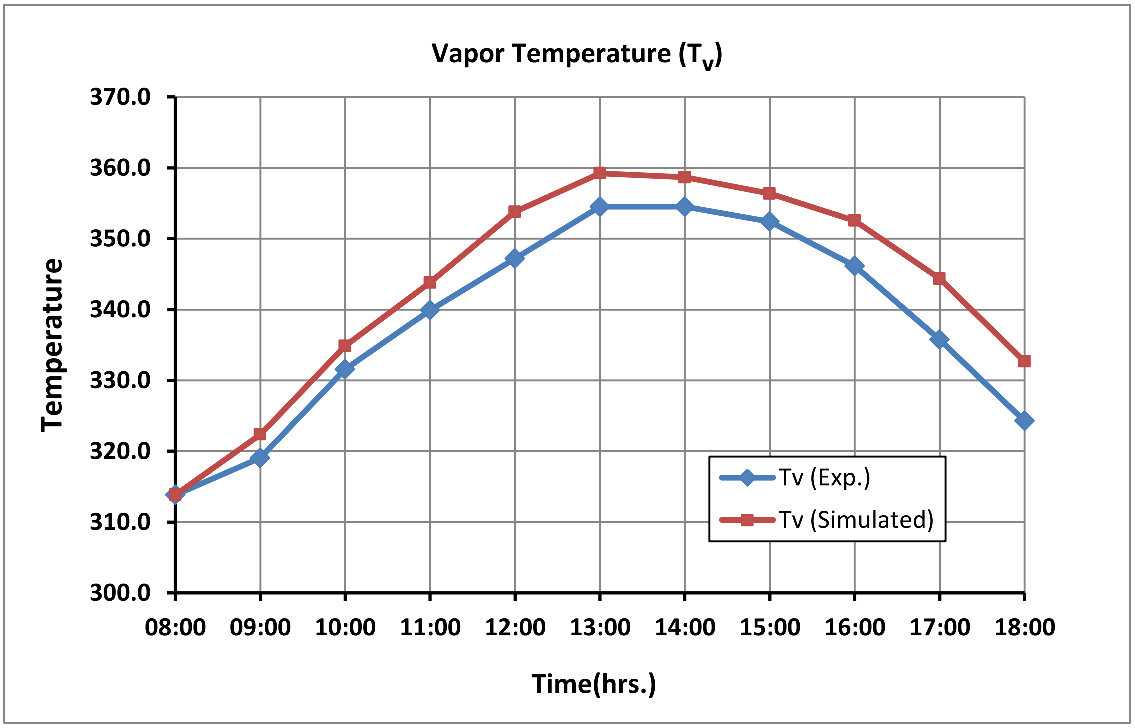

Figure 5.15 Variation of vapor temperature with time

Figure 5.16 shows the comparison of distillate output values from CFD data and from the experimental data. Simulated values are based upon Dunkle's model from CFD data as it gives results with good accuracy for normal temperature range. From the comparison it is clear that the CFD prediction and experimental results are agreeable.

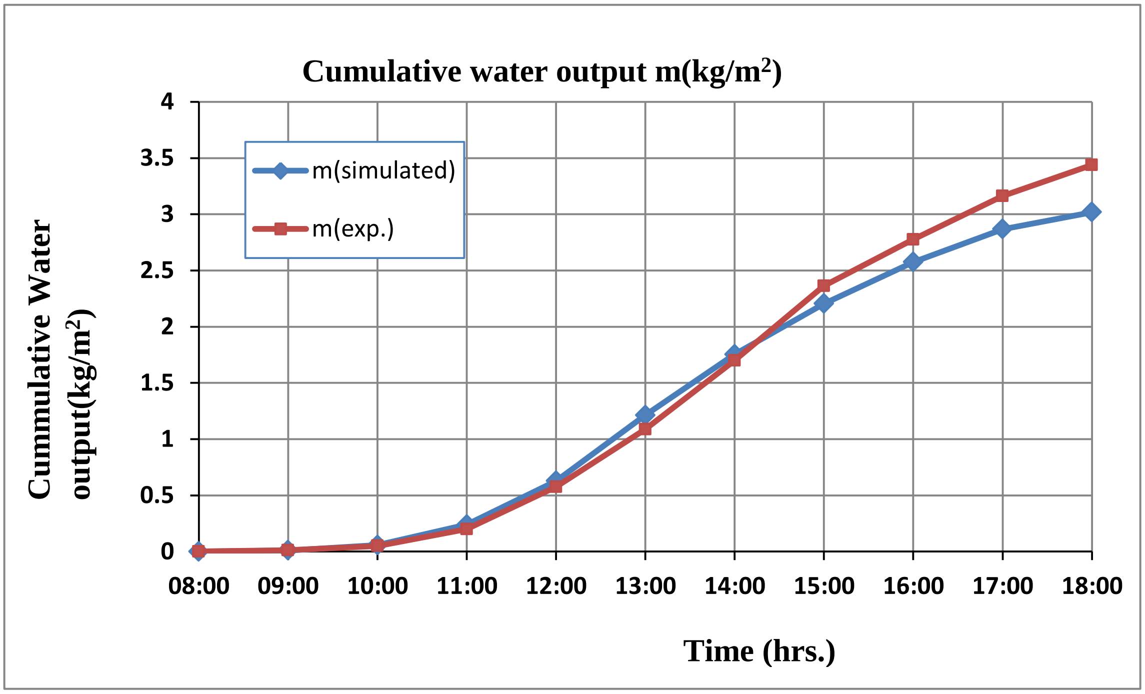

Figure 5.16 Cumulative water output

5.3 Effect of basin water depth on distillate output

There are several factors which affect the distillate output coming out of a solar still. Those include factors such as tilt angle; type of solar still, preheating of basin water, depth of water in the basin, ambient wind velocity and temperature etc. In the present work attempt has been made to investigate the effect of initial quantity of water in the basin on the distillate output of the solar still. Three different quantities of water namely 10 litre, 15 litre and 20 litre have been considered and the effect of changing the depth on distillate output was investigated through CFD simulations.

Figure 5.17 shows the time variation of water temperature and glass temperature along with the distillate output with the initial basin water quantity of 20 litres.

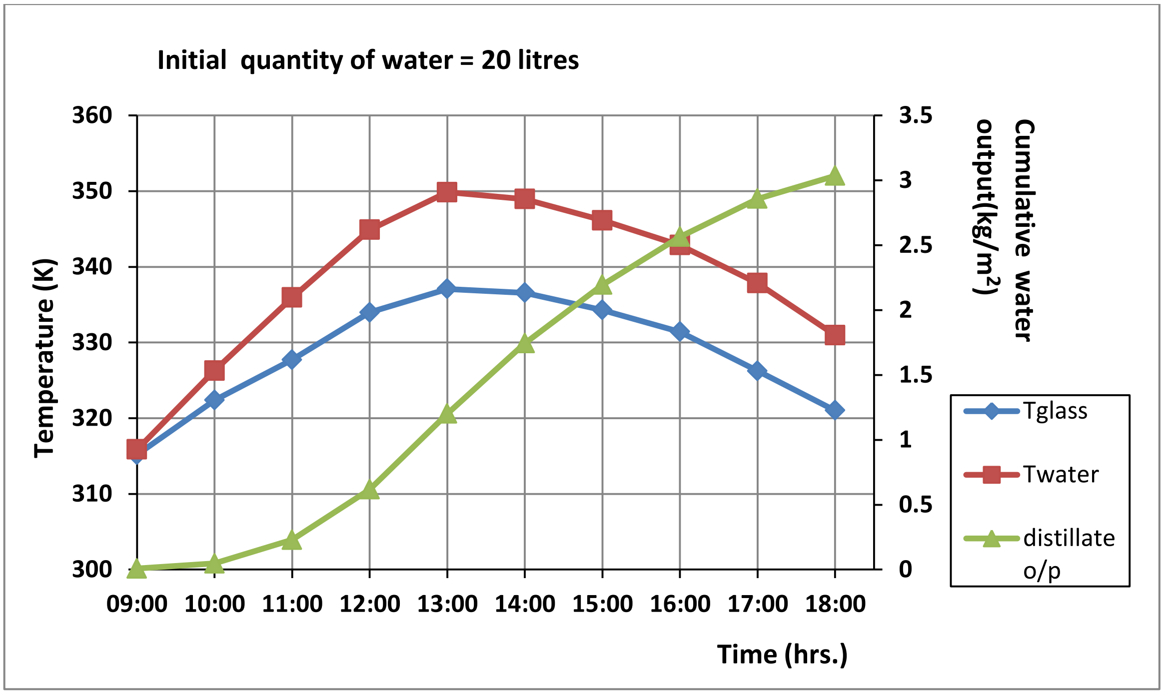

Figure 5.17 Variation of Temperatures and distillate output with 20 litres water

Generally higher values of temperature are attained by the still in which water depth is comparatively lower. This is due to the fact that less mass of water in the basin offers less value of heat capacity which makes the water to gain temperature rapidly. The maximum value of temperature reached by the less mass of basin water is comparatively higher. The higher values of temperature increase the evaporation rate which in turn increases the distillate production rate.

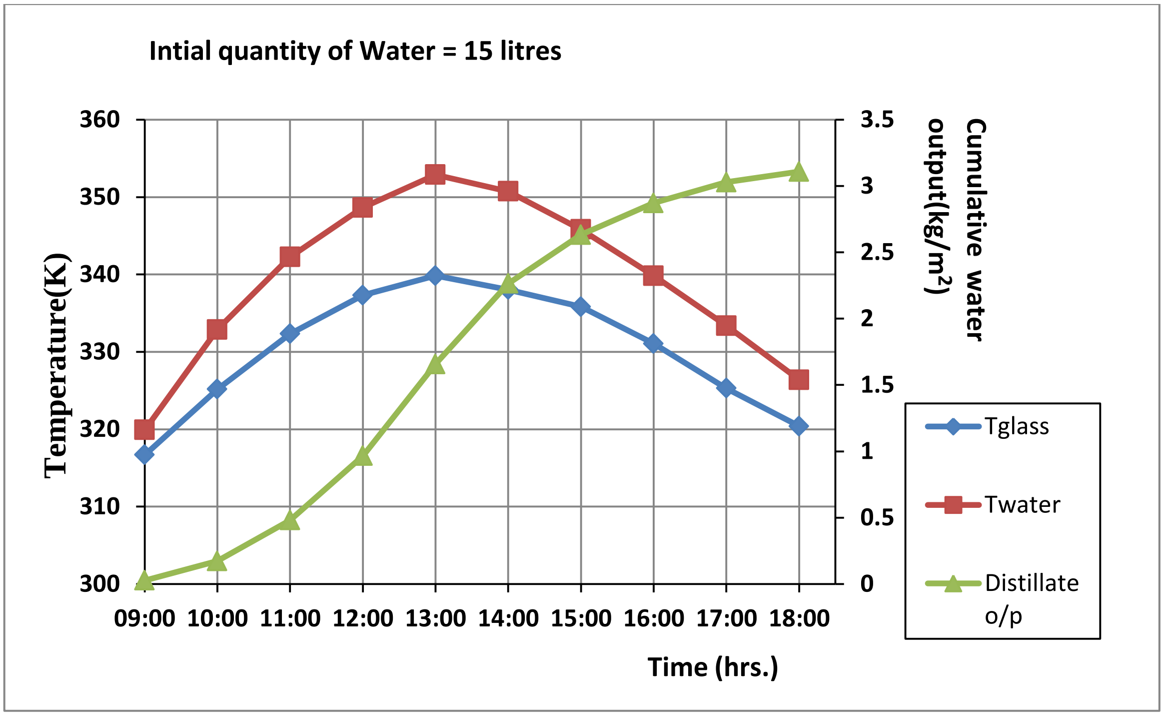

Figure 5.18 Variation of Temperatures and distillate output with 15 litres water

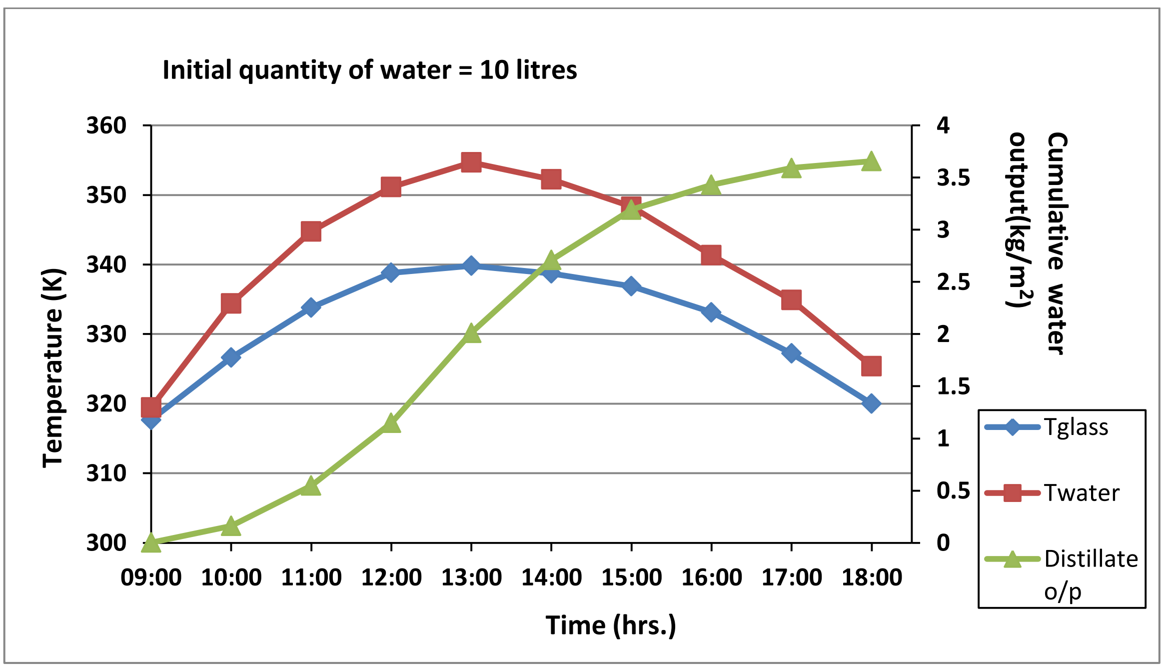

Figure 5.19 Variation of Temperatures and distillate output with 10 litre water

The daily productivity of solar still with 20 litres of basin water is found to be 2648 ml. There has been found a little effect of reducing basin water quantity on the daily yield of solar still. By changing the water quantity in the basin from 20 litres to 10 litres the distillate yield was increased. This happens due to the fact that lesser quantity of water in the basin has lesser heat capacity and the water attains relatively high temperatures. A marginal increase of 6% was found in the daily output of distillate water by reducing the initial quantity of basin water from 20 litres to 10 litres.

CHAPTER 6
CONCLUSION AND FUTURE SCOPE

6.1 Conclusions

The main objective of this study was to develop a 3-D CFD model of a single slope passive solar still and to compare the simulation results with the available experimental results. First the 3-D geometric model of solar still was developed in ANSYS Workbench through design modeler. Then appropriate models were selected for the physical phenomena occurring in the solar still. Material properties and appropriate boundary conditions were defined in Fluent for the simulation of the problem. Simulations were carried for unsteady state conditions with FLUENT solver for 19th May. The simulation results were compared with the experimental results taken on 19th May 2014.

The main conclusions of the study are as follows:-

- The present CFD model is able to simulate the temperature at various points in the solar still. The simulated values of interior temperatures are in agreement with the experimental values.
- The glass temperature follows the similar trend as that of solar radiations. It increases with the solar intensity reaching a maximum value of about 337 K at 13:00 hrs and then it starts decreasing due to the decreasing value of solar intensity
- Although the simulated results follow the same trend as the experimental results yet there is a slight difference between the two temperature data. The difference between the two values is most likely due to the difference in solar intensity for two cases and also because of the fact that simulated values do not take into consideration for natural attenuations that occur in real experiments.

- The rate of distillate water output and evaporative heat transfer coefficient are calculated on the basis of Dunkle's model using CFD data. There is a good agreement between the simulated and experimental data for distillate yield.
- The effect of water depth in the basin on distillate yield was investigated through simulations. The simulations were carried out at three different water quantities in the basin as 10 litres, 15 litres and 20 litres. The distillate output increases with decreasing the basin water depth though not very significant change was found in the value of distillate output. A marginal increase of approximately 6% in the distillate output was found when the quantity of water in the basin was decreased from 20 litres to 10 litres.

6.2 Scope for future work

From the literature review it has been concluded that there has been a little work in CFD modeling of evaporation and condensation phenomena that occurs in the solar still. Based on the experience gained during this present work, following points are suggested which can help in identifying the future scope of this present work:-

- CFD models for more complex designs of solar still such as double slope solar still, wick type solar still and multi effect solar still can be developed and simulated in fluent. Their CFD based performance comparison can be conducted.
- In this study only one parameter i.e. water depth in the basin was considered. There can be various other parameters such as tilt angle of glass cover, absorptivity of the basin water, gap distance between glass cover and bottom of solar still etc. which also affects the distillation yield.
- In order to generalize the effect of various parameters on the performance of solar still, various types of performance correlations can be generated by using the CFD model of solar still.
- CFD analysis of a solar still consisting of a separate condenser section can be conducted in order to enhance the productivity solar still.

- Complex designs of solar still such as still having corrugated basin and still having water flowing over the glass cover can also be modeled by using thin film flows in CFD tools.
- More improved models can be used to accurately simulate the model. For instance DO model for solar radiation and Euler model for multiphase would provide highly accurate results.
- Geometrical optimization of the solar still can be performed by the CFD modeling of solar still.

REFERENCES

[18-1] Sampathkumar K., Arjunan T. V., Pitchandi P., and Senthilkumar P., "Active solar distillation—a detailed review." *Renewable and Sustainable Energy Reviews* 14, (2010): 1503-1526.

[2] Khalifa A.J.N. and Ahmad M. H., "Performance correlations for basin type solar stills." *Desalination* 249 (2009): 24-28.

[3] Kalidasa M. K., Chockalingam Kn. K.S.K., and Srithar K., "Progresses in improving the effectiveness of the single basin passive solar still." *Desalination* 220, (2008): 677-686.

[4] Shankar P. and Kumar S., "Solar Distillation–A Parametric Review."*VSRD International Journal of Mechanical, Automobile and Production Engineering* 2, (2012): 17-33.

[5] Ayoub G. M., Malaeb L., and Saikaly P. E., "Critical variables in the performance of a productivity-enhanced solar still." *Solar Energy* 98 (2013): 472-484.

[6] Badran, O. O. "Experimental study of the enhancement parameters on a single slope solar still productivity." *Desalination* 209, (2007): 136-143.

[7] Kumar S., Dubey A. and Tiwari G. N., "A solar still augmented with an evacuated tube collector in forced mode." *Desalination* 347 (2014): 15-24.

[8] Arunkumar T., Vinothkumar K., Ahsan A., Jayaprakash R., and Kumar S., "Experimental study on various solar still designs." *ISRN Renewable Energy* (2012).

[9] Sahoo, B. B., N. Sahoo, P. Mahanta, L. Borbora, P. Kalita, and U. K. Saha. "Performance assessment of a solar still using blackened surface and thermocol insulation." *Renewable Energy* 33, (2008): 1703-1708.

[10] Rajaseenivasan, T., Elango T., and Murugavel K. K., "Comparative study of double basin and single basin solar stills." *Desalination* 309 (2013): 27-31.

[11] Kalidasa K. M., Chockalingam Kn. K.S.K., and Srithar K., "An experimental study on single basin double slope simulation solar still with thin layer of water in the basin." *Desalination* 220, (2008): 687-693.

[12] Abdullah, A. S. "Improving the performance of stepped solar still." *Desalination*319 (2013): 60-65.

[13] Abdallah S., Badran O. and Abu-Khadr M. M., "Performance evaluation of a modified design of a single slope solar still." *Desalination* 219, no. 1 (2008): 222-230.

[14] Dev R., Sabah A., Wahab A. and Tiwari G. N., "Performance study of the inverted absorber solar still with water depth and total dissolved solid." *Applied energy* 88, (2011): 252-264.

[15] Taghvaei H., Jafarpur K., Estahbanati M.R.K., Feilizadeh M. and Ardekani A. S. "A thorough investigation of the effects of water depth on the performance of active solar stills." *Desalination* 347 (2014): 77-85.

[16] Bhardwaj R., Ten Kortenaar M. V. and Mudde R. F., "Influence of condensation surface on solar distillation." *Desalination* 326 (2013): 37-45.

[17] Kumar S. and Saini R. P., "CFD based performance analysis of a solar air heater duct provided with artificial roughness." *Renewable Energy* 34 (2009): 1285-1291.

[18] Jeon S. S., Kim S. J., and Park G.C., "CFD simulation of condensing vapor bubble using VOF model." *World Acad. Sci., Eng. Technol*60 (2009): 209-215.

[19] Subhashini G. and Nigam K.D.P. “CFD modeling of flow profiles and interfacial phenomena in two-phase flow in pipes” *Chemical Engineering and Processing 45 (2006) 55–65.*

[20] Hitesh P.N., and Shah P. K., "Modeling and verification of single slope solar still using ANSYS-CFX." *International Journal of Energy and Environment* 2, (2011): 985-998.

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

[22] Badusha A. R. and Arjunan T.V., " Performance analysis of single slope solar still." *Desalination* 268 (2011): 103-110.

[23] Xiao G., Xihui W., Mingjiang Ni, Fei W., Weijun Z., Zhongyang L., and Kefa C. "A review on solar stills for brine desalination." *Applied Energy*103 (2013): 642-652.

[24] Kannan R., Selvaganesan C., Vignesh M., Babu B. R., Fuentes M., Vivar M., Skryabin I., and Srithar K. "Solar still with vapor adsorption basin: Performance analysis." *Renewable Energy* 62 (2014): 258-264.

[25] Shatat M., Worall M. and Riffat S. "Opportunities for solar water desalination worldwide: Review." *Sustainable cities and society* 9 (2013): 67-80.

[26] Fluent, Ansys. "Ansys Fluent Theory Guide." *ANSYS Inc., USA* (2011).

[27] Avvannavar S. M., Mani M. and Kumar N. "An integrated assessment of the suitability of domestic solar still as a viable safe water technology for India., " *Environ Eng Manag J* 7, (2008): 667-85.

APPENDICES

Appendix: A. Fluid properties given by FLUENT

Material	Physical properties			
	Density (kg/m^3)	Specific heat (J/kg-K)	Thermal conductivity (W/m-K)	Viscosity (Kg/m-s)
Air	1.225	1006.43	0.0242	1.79E-05
Water-liquid	998.2	4182	0.6	0.001003
Water-vapor	0.5542	2014	0.0261	1.34E-05

APPENDIX : B User Defined Functions

A user defined function (UDF) is sometimes used with the FLUENT solver to enhance the standard features of the code. In this study the UDF is used to define mass transfer phenomena and the source term in the cell zone conditions accounts for the energy source in the still because of phase change i.e. evaporation and condensation.

```
#include"udf.h"
#include"sg_mphase.h"
#define T_SAT 373
#define LAT_HT 1.e3
DEFINE_SOURCE(liq_src,cell,pri_th,dS,eqn)
{
Thread*mix_th,*sec_th;
real m_dot_l;
mix_th=THREAD_SUPER_THREAD(pri_th);
sec_th=THREAD_SUB_THREAD(mix_th,1);
if(C_T(cell,mix_th)>=T_SAT){
m_dot_l=-0.1*C_VOF(cell,pri_th)*C_R(cell,pri_th)*
fabs(C_T(cell,pri_th)-T_SAT)/T_SAT;
dS[eqn]=-0.1*C_R(cell,pri_th)*
fabs(C_T(cell,pri_th)-T_SAT)/T_SAT;
}
else {
m_dot_l=0.1*C_VOF(cell,sec_th)*C_R(cell,sec_th)*
fabs(T_SAT-C_T(cell,mix_th))/T_SAT;
dS[eqn]=0;
}
return m_dot_l;
}
DEFINE_SOURCE(vap_src,cell,sec_th ,dS,eqn)
{
Thread*mix_th,*pri_th;
real m_dot_v;
mix_th=THREAD_SUPER_THREAD(sec_th);
pri_th=THREAD_SUB_THREAD(mix_th,0);
if(C_T(cell,mix_th)>=T_SAT){
m_dot_v=0.1*C_VOF(cell,pri_th)*C_R(cell,pri_th)*
fabs(C_T(cell,mix_th)-T_SAT)/T_SAT;
dS[eqn]=0;
}
else {
m_dot_v=-0.1*C_VOF(cell,sec_th)*C_R(cell,sec_th)*
```

```
fabs(T_SAT-C_T(cell,mix_th))/T_SAT;
dS[eqn]=-0.1*C_R(cell,sec_th)*
fabs(C_T(cell,sec_th)-T_SAT)/T_SAT;
}
return m_dot_v;
}
DEFINE_SOURCE(enrg_src,cell,mix_th,dS,eqn)
{
Thread*pri_th,*sec_th;
real m_dot;
pri_th=THREAD_SUB_THREAD(mix_th,0);
sec_th=THREAD_SUB_THREAD(mix_th,1);
if(C_T(cell,mix_th)>=T_SAT){
m_dot=-0.1*C_VOF(cell,pri_th)*C_R(cell,pri_th)*
fabs(C_T(cell,pri_th)-T_SAT)/T_SAT;
dS[eqn]=-0.1*C_VOF(cell,pri_th)*C_R(cell,pri_th)/T_SAT;
}
else
{
m_dot=0.1*C_VOF(cell,sec_th)*C_R(cell,sec_th)*
fabs(T_SAT-C_T(cell,mix_th))/T_SAT;
dS[eqn]=-0.1*C_VOF(cell,sec_th)*C_R(cell,sec_th)/T_SAT;
}
return LAT_HT*m_dot;
}
```

Appendix : C Experimental Readings available (Taken on 19th May 2014)

Time	T_g	T_v	T_{wa}	$T_{amb.}$	$I (w/m^2)$	Cumulative Output (ml.)
08:00	35.1	40.7	32.8	30.8	225	0.00
09:00	40.0	45.9	39.8	33.8	418	10.0
10:00	45.5	58.4	49.8	34.8	572	40.0
11:00	52.2	66.8	59.7	37.8	694	160.0
12:00	57.5	74.1	67.7	37.8	755	460.0
13:00	60.4	81.4	72.6	38.8	750	870.0
14:00	59.0	81.4	72.6	38.8	702	1360.0
15:00	57.0	79.3	69.7	37.8	570	1890.0
16:00	55.0	73.0	65.7	39.8	418	2220.0
17:00	47.3	62.6	58.7	38.8	225	2530.0
18:00	43.3	51.1	51.7	37.8	67	2750.0

All Temperatures are in °C