2nd National Conference Recent Innovations in Science and Engineering (NC-RISE 17) ISSN: 2321-8169
Volume: 5 Issue: 9 01 – 06

Validation and Computational Performance Analysis of Masonic Solar Still in Summer Climate

P. D. Besekar
M.Tech. (Mechanical), NIT Jamshedpur (Jharkhand), India
E-Mail: psbesekar@gmail.com

Abstract- Supply of drinking water is a major problem in under-developed and in some developing countries. The use of solar energy for desalination purpose was one of the first processes developed for producing fresh water from salt water. The process is based on the use of solar thermal energy to evaporate water thus separating pure water from brine water. The solar stills are used as a good method for obtaining the fresh water for even small scale demand because of economic and technical advantages as it uses the inexpensive technology including the material prices and manufacturing. Hence for increasing the outcome of solar still it is necessary to test the effect of different parameters as condensing cover. Present work aimed at modeling of single slope solar still using ANSYS Fluent V15 to investigate the yield. Three dimensional two phase model is developed for evaporation and condensation process to simulate the temperature distribution of water and gas phase and also the amount of fresh water productivity. Simulation is carried out from 12:00PM to 03:00PM with one hour interval. For each interval various graphs have been plotted and studied for each simulation. It was concluded that for the interval 02:00PM to 03:00PM obtain the higher yield as compared to other interval yield. Simulation results have been compared with the available experimental data it was observed that fresh water production rate and water temperature are in good agreement. Hence Computational Fluid Dynamics is powerful tool in design of solar still and studying effective parameters on the performance.

Keywords: *Desalination, Solar Energy, Thermal Energy, Ansys, Fluent, Computational Fluid Dynamics*

*****

Introduction

The purification of liquids such as water can be done using Solar Still. The impurities present in the liquids can be separated using the process of distillation.Distillation is a simple process. Heat is first added to a liquid to evaporate it and produce a gas or vapor, then heat is removed from the vapor to condense it back to liquid. A solar still works on the greenhouse effect phenomenon. The solar energy which is free is the energy input to this distillation [1].

1.1 Water demand

The four most important uses of water are
1. Drinking
2. Domestic
3. Agricultural
4. Industrial

Ninety-seven percent of the earth's water lies in its oceans. Of the remaining 3%, over 80% is brackish and unfit for consumption leaving just 5% as fresh water. As a result, many people do not have access to plentiful and affordable supplies of potable water. This leads to population concentration around existing water supplies, marginal health conditions, and a generally lower standard of living.

Human beings need 3 to 4 liters of water a day to live and for daily usage it counts approximately equals to 20 liters. Industries require 200 to 400 liters of water as per their needs. Yet some functions can be performed with salty water and a typical requirement for distilled water is 5 liters per person per day. Therefore 2m² of still are needed for each person served [1].

1.2 Water Shortage Problem

The lack of potable water poses a big problem in southern and south-western arid regions of India. The underground water, where exists, is usually brackish and cannot be used as it is for drinking purposes. As India lies in the region of high solar potential, we can have many advantages of the solar still setup. The most economical and easy way to accomplish this objective is using solar still. Fresh water scarcity is a pressing problem that progressively affects more and more region on the planet due to the continuous increase in world population, changes in lifestyle and increasing contamination of existing natural fresh water resources.

1.3 Solution to the Water Problem

The demand for a steady, economical supply of water is constantly increasing around the world. Often it does not match the available supply. It does not seem possible that supply will equal demand in the near future. Therefore, the development and management of sound water resources will be a constant challenge. In many countries, water policy will be an essential ingredient of economic policy. There are many solutions to the water problem, including control of water consumption, conservation, improved distribution and storage, reclamation, purification and reuse, crops that use less water, tapping of new sources, etc. Desalination is seriously considered only when all the other possibilities have been ruled out Seawater desalination plants have been constructed in many countries, especially the arid Middle East, only because there were no other available alternatives. The objective of desalination is to provide water with salinity below 500 ppm. The major problems associated with desalination have been very high capital and operating costs. Over the past several years, the cost of desalting has gone down but it is still quite high. It still cannot compete with the cost of natural fresh water, which has the advantage that it requires minimal treatment to make it potable.

Though many methods have been proposed to desalt saline waters, only a few have been developed to commercial viability. The majority of commercial desalination processes have been perfected over the past 50 years. The applicability of any process depends on the amount of salts contained in the available feed water and on process economics.

1.4 Modeling Procedure in Computational Fluid Dynamics

Fluid dynamics deals with the dynamic behavior of fluids and its mathematical interpretation is called as Computational Fluid Dynamics. Fluid dynamics is governed by sets of partial differential equations, which in most cases are difficult or rather impossible to obtain analytical solution.Computational Fluid Dynamics is a computational technology that enables the study of dynamics of things that flow.

The Physical aspects of any fluid flow are governed by three fundamental principles: Mass is conserved; Newton's second law and Energy is conserved, mathematically they can be expressed as partial differential equations. Computational Fluid Dynamics is the science of determining a numerical solution to the governing equations of fluid flow.

Computational Fluid Dynamics thus provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of Mathematical modeling (partial differential equations) Numerical methods (discretization and solution techniques).

Computational Fluid Dynamics enables scientists and engineers to perform ‘numerical experiments’i.e. Computer simulations in a ‘virtual flow laboratory’ real experiment Computational Fluid Dynamics simulation.

The procedure for the Computational Fluid Dynamics analysis in FLUENT follows the simple steps below:

1. The model used for the analysis is drawn in, NX and meshed in the ICEM- Computational Fluid Dynamics software, which is the compatible modeling software for FLUENT. All the files for the geometry and meshing of the model are saved as mesh or grid file.
2. Next, in FLUENT, the saved mesh or grid file of the model is read, checked and scaled for the required working unit.
3. The model is defined for the type of solver and boundary conditions to be used. The model is defined according to the type of analysis required in the research project.
4. The model is solved by setting the required parameters in the solution panel and then iterated for convergence.
5. Results can be obtained from the graphic display and report in FLUENT. Results can be displayed in terms of contour, velocity vector, and particle track and path line. Any calculation required can be performed in FLUENT also.
6. Finally, the results and all the data can be saved for future references by writing the files.

The schematic diagram of experimental set up of Masonic solar still is as shown in Fig.1 Experimental set up consist of Basin tank which is basic Masonic solar still and it has effective area of 1 m^2. The still is made of Bricks, tiles, Cement concrete. It has a top cover of transparent glass (i.e. Ordinary glass), and the interior surface of its basin is black colored Rexene to enable absorption of solar energy. The whole assembly was air tight made with help of Rexene strips and sealed with silicon paste.

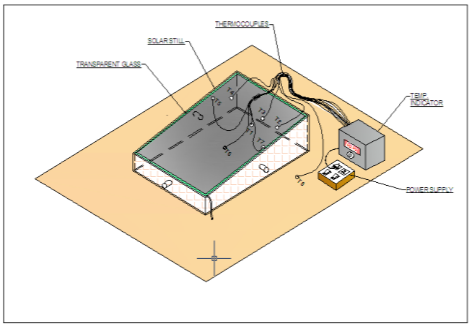

Fig: 1Schematic Diagram of Masonic Solar Still

In this diagram shows the different component and how it’s assembled. Here use a eight channel thermocouple for measuring the different temperature i.e. water basin temperature (T6), inner glass temperature (T5&7), Dry bulb temperature (T1), atmospheric temperature (T8) and outside glass temperature (T2, T3, T4). A plain transparent glass (i.e. Ordinary) of 4 mm thickness was used at the top, with an inclination of 18.5° from horizontal to ease the flow of condensate along the transparent inside surface of glass. The low thickness of glass transmits up to 98% of solar radiation. The solar still efficiency depends on the properties of the transparent glass.

II. METHODOLOGY

2.1 Simulation Method

Here in this chapter starting from the development of two phase three dimensional model of solar still with assumptions each step for carrying out simulation is explained. Size of geometry, meshing, boundary condition applied on the simulation model and simulation procedure has been discussed here with an enhanced contrast.

2.1.1 Assumptions

1. There is no vapour leakage from the solar still.
2. All the walls are adiabatic, hence no heat loss.
3. There is no temperature gradient across the glass cover of solar still.

2.1.2 Mathematical Model

A two-phase model was developed in the volume of fluid for liquid water and mixture of air and water vapor system at quasi steady state condition and adaptive time step has taken. So only surface evaporation of liquid occurs and their interface should be considered for modeling. For each phase the time and volume-average continuity, energy and mass equations are numerically solved.

2.2 Governing Equations in Computational Fluid Dynamics

There are mainly three equations we solve in computational fluid dynamics problem. They are Continuity equation, Momentum equation (Navier Stokes equation) and Energy equation. The flow of most fluids may be analyzed mathematically by the use of two equations. Basically, *Continuity Equation* states that the mass of fluid entering the fixed control volume is equal to the mass of liquid leaving. It is thus a "mass balance" requirement posed in mathematical form, and is a scalar equation. While, the*Momentum Equation*, or *Navier-Stokes Equation*may be thought of as a "momentum balance". The Navier-Stokes equations are vector equations, meaning that there is a separate equation for each of the coordinate directions (usually three) [8].

2.3 Steps involved in solving problem

1. First create the grid of appropriate dimensions and with appropriate step length to specify the problem domain in ICEM Computational Fluid Dynamics
2. Create geometries like Vertices at appropriate grid points.
3. Create lines joining two vertices.
4. Create Areas selecting all the lines.
5. Create Boundary Mesh around the cylinder.
6. Create Face Mesh to rest of the model.
7. Give the Boundary Conditions for entire domain.
8. Save it and export it to mesh file.
9. Read the file in FLUENT and check the mesh and scale the model.
10. Enter values for boundary conditions, operating conditions etc.
11. Selecting the appropriate solver to solve the problem.
12. Solve the problem by initializing from velocity inlet and specifying the number of iterations.

2.4 Simulation

For the simulation of condensation and evaporation process inside the distillation unit, Computational Fluid Dynamics simulation software ANSYS Fluent V15 is used. ANSYS Fluent V15 is a fluid analysis and simulation software which combines CAD automatic Meshing and fast solution algorithms, which include Pre-processing, solving, Post processing. Preprocessor include modeling (CAD), meshing, applying boundary conditions, Solver which is based on finite volume method Include applying the solution algorithms and iterating and forming the results. Post processor includes displaying the results, forming temperature diagrams, contours graphs on display screen.

2.5 Methodology

2.5.1 Geometric model

Geometric modal of solar still is created using ANSYS CAD module and this geometric modal is imported to the ANSYS meshing module to generate the meshing of the solar still. Geometry is shown in figures 2.

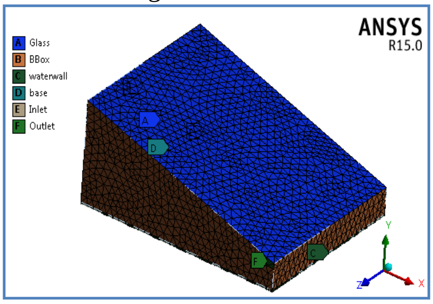

Fig: 2 3D model of Masonic solar still

2.5.2 Meshing Details

Geometry is imported to ANSYS meshing module and meshing is done. Unstructured mesh of type tetrahedral is used. A mesh independence test was conducted to ensure the results obtained are accurate and reliable.

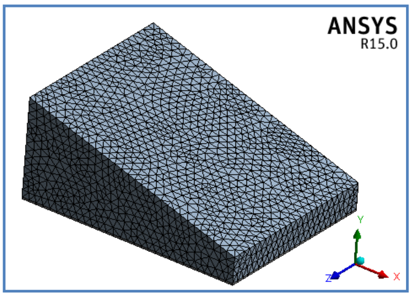

Fig: 3 3D unstructured mesh for solar still

2.5.3 Boundary Conditions

Mesh files of solar still are then imported to the Fluent Pre. In Fluent pre physics and boundary conditions are applied on the domain is explained to solve the continuity and momentum equation. A two phase domain is created in the framework for liquid water and mixture of water vapour and air. Considering the effect of buoyancy, evaporation process is modeled as laminar at quasi steady state. A distinct interface between vapor and liquid phases exists, hence both phases are continuous. To transfer heat, two resistance model is taken zero resistance model is taken for gas phase and heat transfer coefficient for water phase. It was assumed that bottom temperature is equal to water bath temperature. Distillate collector temperature is equal to glass temperature. For drop formation on the condensing cover adhesion forces are taken. All sides are assumed adiabatic. No slip
Boundary condition is specified for liquid phase and free slip boundary condition is specified for vapour phase.

2.5.4 Material Definition

To set up the domain for 2-phase fluid (Multiphase) analysis, First of all fluid material have defined
Two types of materials are as follows:
1. Liquid water
2. Water vapour

ISSN: 2321-8169
01 – 06

In ANSYS Fluent V15 material named gas mixture was prepared by combining the water vapour and air. They are defined by using the variable composition mixture. The thermal properties of water and water vapour mixture are given in table.

Table no 1: Thermal properties of water

Density	988.2[kg/m^3]
Molar Mass	18.0152 [kg/ kmol]
Specific Heat Capacity	4182 [J/kg-K]
Viscosity	0.001003[kg/m-s]
Thermal Conductivity	0.6 [W /mK]

Table no 2: Thermal properties of water vapour

Density	0.5542[kg/m^3]
Molar Mass	18.0152 [kg/ kmol]
Specific Heat Capacity	300-1000 [J/kg-K]
Viscosity	1.345[kg/m-s]
Thermal Conductivity	0.0261[W /mK]

2.5.5 Fluent Solver

Fluent solver performs the iteration process when the iteration process comes to the end of convergence as shown in Fig 4 it provides the results to the Fluent Processor.

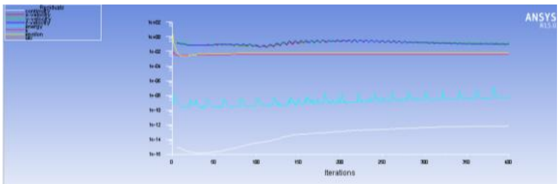

Fig: 4 Convergence graph of fluid flow simulation

III. RESULTS AND DISCUSSIONS

3.1 Computational Fluid Dynamic Simulations

The performance of a solar still depends upon the glass cover angle, depth of water, fabrication materials, and temperature of water in the basin and insulation thickness which could be modified for improving the performance. So Computational fluid dynamic simulation approach is adopted to analyze the effect of temperature on yield of a solar still. In evaporation water phase change takes place which is very difficult to model mathematically. The complicated thermal relationship involved in the phase change of water from liquid to gas phase is accurately solved by using Computational fluid dynamic. The numerical analysis using ANSYS Fluent V15 is used to determine the temperature distribution. In the present work, following the simulation methodology and utilizing the boundary conditions as mentioned in detail in Chapter 5, simulations were completed to obtain the following sets of results:

1. Validate the experimental yield with simulated yield.
2. Vapour temperature distribution inside the solar still.
3. Water temperature distribution inside the solar still.

3.1.1Temperature Distribution for basin water

To represent results of temperature distribution in the form of 3-D colored contour plots the images have been taken for each individual simulation and few of them as sample representation are shown in Figure 5-7. Red region in the contour plots represent highest temperature and blue region represent the lowest temperature in the domain it is clearly seen in the figure due to large temperature difference between glass and water temperature water condenses on the glass.

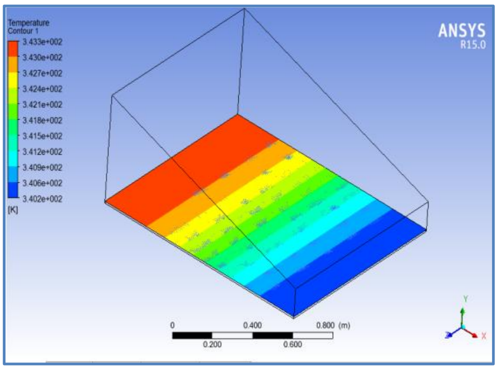

Fig: 5 Temperature of basin water

3.1.1.1Simulation for 12:00 PM to 01:00 PM

Fig.6 shows the results of output water yield by simulation for one hour from 12:00 PM to 01:00 PM. The water in the still starts warming due to solar radiation incident on glass surface. After some time water in the system start to vaporize. The temperature difference between water and glass leads to condensate the vapours. For this interval maximum temperature achieved is 334.7°K and minimum is 331.7°K. From the simulation it is observed that for this interval the mass flow rate is 4.66*10^{-5} kg/sec i.e. 167ml/hour.

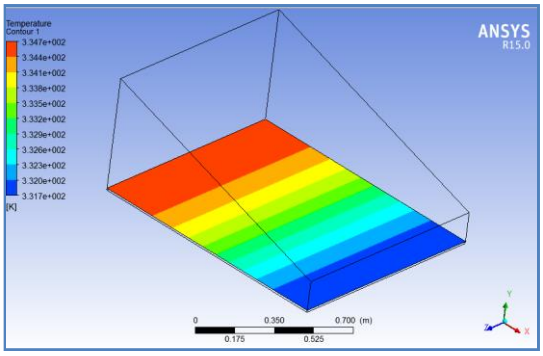

Fig. 6 Water temperature inside the solar still for 12:00PM-01:00PM interval

3.1.1.2Simulation for 1:00PM to 2:00PM

Fig.7 shows the results of output water yield by simulation for one hour from 01:00 PM to 02:00 PM. The water in the still starts warming due to solar radiation incident on glass surface. For this interval maximum temperature achieved is 346.1°K and minimum is 342.4°K. From the simulation it is observed that for this interval the mass flow rate is 5.521*10^{-5} kg/sec i.e. 198ml/hour.

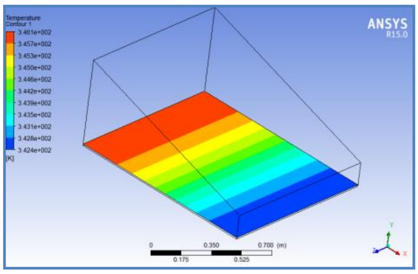

Fig: 7 Water temperatures inside the solar still for 01:00pm-02:00pm interval

3.1.1.3Simulation for 02:00 PM to 03:00 PM

Fig.8 shows the results of output water yield by simulation for one hour from 02:00 PM to 03:00 PM. The water in the still starts warming due to solar radiation incident on glass surface. For this interval maximum temperature achieved is $343.4°K$ and minimum is $340.3°K$. From the simulation it is observed that for this interval the mass flow rate is $6.211*10^{-5}$ kg/sec i.e. 223ml/hour.

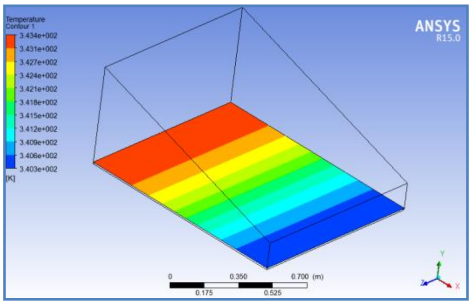

Fig: 8 Water temperature inside the solar still for 02:00 PM to 03:00 PM interval.

3.1.2 Temperature Distribution for glass surface

Solar radiations are incident on a glass surface temperature in the start increasing and it is maximum at the top and it decreases towards the bottom. To represent results of temperature distribution in the form of 3-D colored contour plots the images have been taken for each individual simulation and few of them as sample representation are shown in Figure 9. Red region in the contour plots represent highest temperature and blue region represent the lowest temperature in the domain it is clearly seen in the figure due to large temperature difference between glass and water temperature water condenses on the glass.

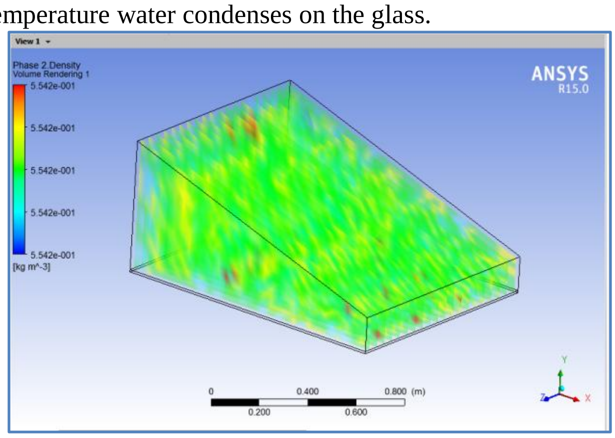

Fig: 9 Water evaporating and stalling on Glass Face

3.2 Experimental Validation

3.2.1 Water Temperature

From the above results water temperature predicted by Computational fluid dynamic simulation is compared with available experimental data of masonic solar still for shown in figure 10 .This Fig has been drawn with Water temperature as ordinate and time interval as abscissa. Water temperature is in good agreement with experimental data.

Table no 4: Comparative Table for Basin Temperature

Sr. No.	Interval	Experimental Average basin Temperature in K	Simulated Average basin Temperature in K
1	12:00PM-01:00PM	321	333
2	01:00PM-02:00PM	324	344
3	02:00PM-03:00PM	322	341

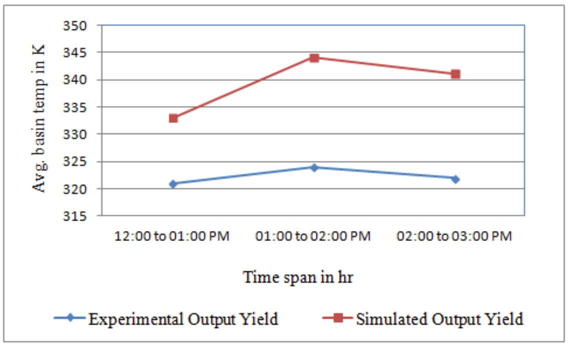

Fig: 10 Water temperature predicted by the Computational fluid dynamic simulation and experimental data

3.2.2 Production of water

When the salinated water is heated water gets evaporated and converted into water vapour which is condensed to obtain potable water in liquid state. ANSYS gives the mass flow of gas and water in Kg/sec which is divided by evaporation area to obtain the amount of fresh water produced. It was assumed that the amount of water evaporated is equal to the rate of water production Kg/m2 sec. In other words the amount of water evaporated is equal to the amount of water condensed. The same amount of water is collected in distillate channel. From simulation the amount of fresh water produced is calculated. In the starting water production rate is low as the basin temperature is low. Figure 11 shows the results of simulation run and experimental data from12:00pm to 03:00pm with one hour interval and figure 12 for number of days with total yield. In the figure 11 it is noted that as the process begins water in the basin is heated by solar radiation. Water starts warming up. Gradually the still space saturates with vapour and fresh water production rate is maximum for last interval. It has been found that water production rate increases with increase in temperature difference. This is due to the fact that, at higher temperature evaporation rate is high. The amount of distillate output received will be higher for the higher temperature of evaporative surface and also for lower temperature of condensing surface.In other words, the higher value of evaporative surface temperature and lower value of condensing surface temperature, both leads to the rise in distillate output. A little consideration reveals that both of this increases the temperature difference between evaporative and condensing surfaces. It plays an important role in optimizing the yield. The average error for output yield is 4.5% i.e. from Computational fluid dynamic simulation 4.5% higher yield than experimental data is obtained.

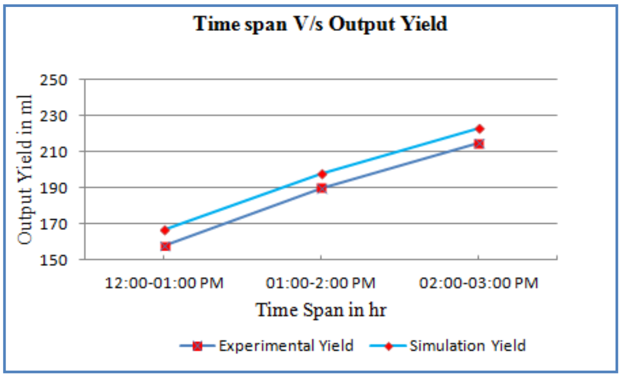

Fig: 11 Fresh water production from experimental data and simulation result for one day on hourly basis

Table no 6: Comparative Table of Yield for the days

Sr. No.	Date	Experimental Average Yield (in ml)	Simulated Yield (in ml)
1	12/4/2016	580	588
2	13/4/2016	510	520
3	14/4/2016	520	528
4	15/4/2016	480	491
5	16/4/2016	680	687
6	17/4/2016	505	515
7	18/4/2016	580	588
8	19/4/2016	525	532
9	20/4/2016	510	520

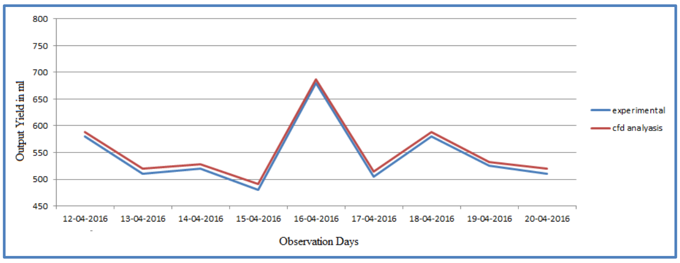

Fig: 12 Fresh water production from experimental data and simulation result for number of days

IV. CONCLUSIONS

4.1 Flow of Work Done

First of all simulation work is done on ANSYS Fluent V15. Simulation of evaporation process is done on a single slope solar still. Different sets of simulation are done for different intervals. The behavior of phase change and temperature distribution is observed due to evaporation. The temperature of water obtained by Fluent and mass yield is compared with the available experimental data.

4.1.1 Experimental validation

It was observed that the results produced by Computational Fluid Dynamics simulation are good in agreement with experimental data. It was concluded that for the interval 02:00PM to 03:00PM obtain the higher yield as compared to other interval yield. Also it is concluded that yield of solar still i.e. distillate output of solar still is directly proportional to the condensation rate.

V. FUTURE SCOPES

There were several improvements that could be implemented to improve the design and Performance, but this was not possible to do so during the course of the project due to time restrictions. These suggestions are recommended for any future work to be conducted on the solar still for desalination purposes:

1. Computer simulation can be used for simulating evaporation process through new solar still designs. Designs of solar still can be improved further by changing different cross-sections like rectangular or cylindrical or by changing the shapes of condensing cover.
2. A Computational fluid dynamic analysis should be conducted on the condensation process by considering separate condenser.
3. Designs of solar still can be improved further by changing different inclinations.

REFERENCES

[1] Palak Patel," A Review to "Increase the Performance of Solar Still: Make It Multi-Layer Absorber", International Journal on Recent and Innovation Trends in Computing and Communication ISSN: 2321-8169 Volume: 2 Issue: 2 173 – 177

[2] Hassan E.S. Fath" Development in Simple Solar Stills", Renewable Energy Systems and Desalination – Vol. iii.

[3] Said Alforjani Said 1, "Flexible Design and Operation of Multi-Stage Flash Desalination Process Subject to Variable Fouling and Variable Freshwater Demand", Processes 2013, 1, 279-295; doi:10.3390/pr1030279

[4] Al-Shammiri M., Safar M(1999). Multi-effect distillation plants: state of the art. Desalination , 126:45- 59.

[5] Delyannis E. (2003), Historic background of desalination and renewable energies. Solar Energy 75(5), Elsevier pp. 357-66.

[6] Dr. S. Shanmugan, "Experimental investigation of various energy absorbing materials on performance of single slope single basin solar still With Hot water", International Journal of Innovative Research in Science, Engineering and Technology (An ISO 3297: 2007 Certified Organization) Vol. 2, Issue 12, December 2013

[7] WangdaZuo," Introduction of Computational Fluid Dynamics", JASS 05, St. Petersburg .

[8] S Kumar and GN Tiwari (1996) Estimation of convective mass transfer in solar distillation system, Solar Energy 57, 459-4