Numerical and Experimental Investigation of Scheffler Concentrator Receivers for Steam Generation Rate Under Different Operating Conditions

Prof. Anita A. Nene*, Dr. S. Ramachandran**

*Department of Mechanical, Research scholar, Sathyabama University, Jeppiaar Nagar, Chennai – 600119, India
**Department of Mechanical, Faculty of Engineering, Sathyabama University, Jeppiaar Nagar, Chennai – 600119, India
anita.nene@mitpune.edu.in, aishram2006@gmail.com.

‡ Corresponding author;First Author,New Ajanta Avenue,3B/A2/7,Paud Road, Kothrud, Pune-411038,

Tel:+91 9657712411 Email ID:anita.nene@mitpune.edu.in,

Received: 26.04.2017 Accepted:09.09.2017

Abstract -Concentrated Solar Technology (CST) is a device which can concentrate solar radiation using mirrors or lenses to produce temperatures in the range of 100^0C to 450^0C or more. Scheffler solar concentrator with fixed focus is one such important CST system. Receiver which contains a working fluid like water is placed at the fixed focus of Scheffler concentrator to receive solar radiations. Significant efforts are being made to analyse, characterise and improve performance of receivers since they have significant impact on overall efficiency of the system. In the subject work Scheffler solar concentrator of $2.7m^2$ area with receivers of Cylindrical and Conical shape having 8 litres capacity were chosen. The scope of study was to map performance of both the receivers under General operating condition, with pre heated water to reduce thermal inertia and introducing a tilt or inclination to the receiver. The special emphasis was on studying influence of these conditions on temperature distribution in the focus region and steam generation rate which finally influence efficiency. Numerical analysis using Fluent-15 CFD package and experimental investigation were performed. Numerical analysis using CFD has helped to map changing temperature regions on the receiver under different conditions and thus provide rationale for influence on efficiency. Cylindrical receiver performed better in General operating condition and Pre heated water, While Tilt improved steam generation rate in conical. Preheating of water gave maximum impact on steam generation rate .The result shows good agreement between numerical and experimental values of mass flow rate of steam with maximum deviation 9 % and 10.21 % for cylindrical and conical receiver respectively and thus validating Numerical method. While CFD FLUENT package has been utilised by other researchers for computing Convective losses, the subject work is directed to mapping temperature distribution and Steam generation rate and hence directly the overall output parameter of solar system and thus easy link to efficiency. For any design of end application of solar system such as process heating its important to know steam generation rate rather than losses and hence the subject work has more practical utility. Besides it creates basis for further efficiency improvements by more changes and estimating their implication.

Keywords – Scheffler concentrator, fixed focus, Receiver, thermal efficiency, CFD

1. Introduction

Scheffler concentrator or collector is used to concentrate solar rays to a receiver positioned at focus. Receiver contains fluid which is heated due to incident concentrated solar flux. Reflector of Scheffler concentrator has efficiency in range of 85-90% while total system efficiency is in range of 30-45%, making receiver a bottleneck for heat flow rate and overall efficiency of system. While the coefficient of absorption of receiver decides amount of heat absorbed, the losses due to radiation & conduction start to rise with temperature of receiver. However the convective losses which are dependent on many variables including wind flow on the surface of receiver are difficult to estimate and build into predictable model. Many efforts are being made to study the losses in order to improve efficiency of receiver and thereby of the total system. [14][16][17[18]. A step towards such effort is to experimentally analyze the performance of receivers under various conditions. However such experimentation becomes an exhaustive exercise and time consuming. Hence various efforts are made to build mathematical models validated by using experimental data. Once such model is built it can become a predictive tool to forecast performance in any new condition or change. Since CFD is such powerful tool available to analyse the fluid flows in the receiver and to map the temperature distribution across the receiver face, it has been thought to be utilised to estimate steam generation rate. From the point of view of any application and user point of view its important to speak efficiency in terms of Steam generation rate and improvement thereupon rather than just in the language of loss reduction percentage .This work directly maps steam generation rate and there by efficiency and hence a new direction.

1.1 Related research

Steam generation capacity of any receiver depends upon the input heat flux and temperature distribution on its surface. CFD simulation tool is used to study this temperature distribution and heat flow from external surface of receiver to inner fluid. Experimental work co-related with Numerical studies on different concentrators and types of receivers is creating predictable validated basis to build reliable model.

Experimental and numerical study of steady state convective losses occurring from a downward facing cylindrical cavity receiver was carried out. Experiments were conducted for fluid inlet temperatures between 50ºC and 75ºC and for receiver inclination angles of 0º (sideways facing cavity), 30º, 45º, 60º and 90º (vertically downward facing receiver). Numerical study was performed for fluid inlet temperatures between 50ºC and 300ºC and receiver inclinations of 0 º, 45 º and 90 º using the Fluent CFD software. The experimental and numerical convective loss estimations agree reasonably well with a maximum deviation of about 14%. It was found that the convective loss increases with mean receiver temperature and decreases with increase in receiver inclination. Nusselt number correlations were proposed for two receiver fluid inlet temperature ranges 50ºC–75ºC and 100ºC –300ºC, based on the experimental and predicted data respectively as in [1]. Convective heat transfer from a body in a virtual wind tunnel was carried out. Air velocity, turbulence intensity and orientation of the body affects convective heat transfer. Some studies have been found to quantify the relationship of airspeed and convective heat transfer from the body. The study was performed by using CFD as in [2] [19] [20]. A 2D numerical analysis was carried out using Fluent 6.3.26(Fluent Inc.)CFD package, on a cavity receiver to study the temperature distribution and losses from the receiver in dry condition during solar period. The non-uniform incident radiation distribution is used as boundary condition for this analysis. Finally CFD results were compared with experimental results as in [3]. The numerical study of natural convection loss occurring from cylindrical solar cavity receivers was performed. Fluent CFD software was used for the analysis of the receivers. In this study the receiver tubes within the cylindrical cavity are modeled as a helical coil similar to those existing in actual systems. The flow of the working fluid within the helical coil is also modeled. The simulations were performed for fluid inlet temperatures of 150◦C and 250◦C and for receiver inclination angles of 0º (sideways-facing cavity), 30º, 45º, 60º, and 90º (vertically downward facing receiver). It is found that the convective loss increases with increasing mean fluid temperature and decreases with increase in receiver inclination A Nusselt number correlation involving Rayleigh numbers, receiver inclinations, and opening ratios is proposed for the convective losses as in [4].Model for a conventional solar distillation system was developed and Experimental work was compared with simulation. Temperature distribution over the glass is verified with Computational Fluid Dynamic using Ansys as in [5]. CFD analysis was performed using experimental data to achieve better efficiency of solar dryer. CFD tool helps to study how the air temperature varies from inlet to outlet as well as throughout the day and also shows the contour of the temperature and velocity inside the dryer as in [6] [21].Computational fluid dynamics technique was used for performance analysis of flat plate collector to improve the thermal performance using a novel cost effective enhanced heat transfer technique. The results are coupled by the findings of simplistic computational fluid dynamics (CFD) designs as in [9]. The use of a validated CFD program (FLUENT) and a solar simulator, for designing a solar water-heater for passive cooling and heating system introduced for buildings in North Africa. CFD transient simulations were carried out using a small time-step of 10 s and a set of one body-fitted computational grid (1770-4740 nodes). FLUENT results were then verified against indoor testing employing a solar simulator as in [10]. Numerical three-dimensional studies of the natural convection and radiation heat loss from cavity receiver of different shapes with and without mouth-blockage have been investigated under isothermal wall condition. Convective heat loss was found to decrease for cavities having mouth blockage created by reducing aperture area, whereas it enhances when mouth blockages are introduced by increasing the cavity dimensions and keeping the same aperture area. Convective loss is characterized by using the convective zone area. Conical cavity yields the lowest convective loss whereas hetro-conical cavity gives the highest convective loss among different shapes investigated. Radiative loss is independent of cavity inclination and was found to be nearly constant for all cavity shapes and cavity configurations as in [11].

2. Scheffler Concentrator System Set Up and Experimental Investigation

2.1 Constructional Details

The Scheffler concentrator used in the subject experimentation is shown in Fig 1 having surface area

$2.7m^2$ with major axis 2200mm and minor axis 830mm. Sun tracking is with manual system with two degrees of freedom. Total 236 mirrors each of 10cm x 9cm are assembled on parabolic steel frame, which direct parallel solar rays aligned to axis of parabola to a single point called focal point where receiver is positioned. [13] [15].

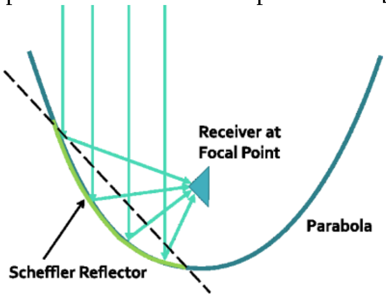


Fig 1.Scheffler Concentrator

2.2 Receiver Configuration

The subject study considers two receiver configurations namely Cylindrical and Conical**.**

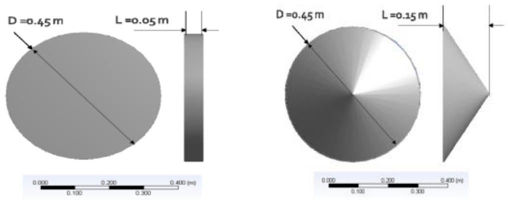


Fig 2.Cylindrical and conical receiver dimensions

Figure 2. Shows Computer Aided Design (CAD) model for cylindrical and conical receiver used in the experimentation. Both receivers have same diameter as 0.45 m. The axial length was adjusted to get the same volume capacity of 8Litre for both receivers. Receiver material chosen was mild steel and surfaces black painted having absorptivity as 0.91. Delta Polyurethane generally known as PUF material of 50 mm thickness was used for insulation on the side surface.

2.3 .Experimental Investigation

Experimental study was done for both receivers under following varying conditions [22] [23] [24]

1. Normal position of receivers herein called "General condition "
2. Initial heating – Water inlet temperature was raised from normal 25^0 to 50^0
3. Tilting the receivers- An optimum condition of 45^0 was found to be the best for reducing convective losses.

Steps while conducting experiment.

1. Adjusting the focus of Scheffler concentrator according to sun angle.
2. On reaching desired pressure of 1 bar , same is maintained by releasing excess steam through pressure relief valve situated on the top of receiver with no adjustment in scheffler concentrator.
3. After getting the desired pressure readings were taken.
4. Readings include- mass flow rate of steam, steam pressure, Air temperature, Global radiation, Diffuser radiation, Direct radiation and Wind speed, time to reach 1 bar.
5. Readings were taken for every 15 mins. (Total fifty two readings were taken during experimentation)
6. After reaching 1bar steam pressure at steady state condition , the Receiver water was collected to estimate mass flow rate of steam.

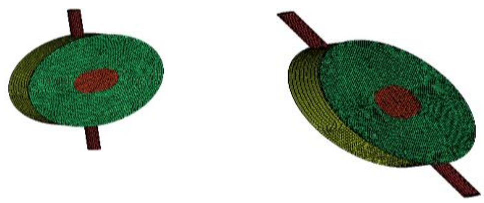
3. Numerical Investigation using Computational Fluid Dynamics (CFD)

In this case 3D numerical analysis is performed, using Fluent-15 CFD package to study the temperature distribution on receiver surface and mass of evaporated water quantity. Numerical value of mass of evaporated quantity of water gives a mass flow rate of steam; hence numerical thermal efficiency can be calculated. The Navier-Stokes equations are the basic governing equations for a viscous, heat conducting fluid. It is a vector equation obtained by applying Newton's Law of Motion to a fluid element and is also called the momentum equation.

It is supplemented by the mass conservation equation, also called continuity equation and the energy equations in [7],[12]. The numerical CFD simulation procedure consists of following steps.

3.1 Pre Processing: Creating Computational domain and grid

As an initial step for the numerical analysis the geometrical model were created. The modelling and meshing part was done using ICEM CFD software.



a) No inclination to Horizontal b) 45° Tilt to Horizontal

Fig 3: Meshing Model for Cylindrical Receiver

The geometry of receiver is straight and inclined to 45° with horizontal plane as per experimental setup. An unstructured mesh with Tetrahedron and prism element is used for meshing of receiver geometry as shown in fig.3 and fig.4

(a)No inclination to Horizontal (b)45° Tilt to Horizontal

Fig .4: Meshing Model for Conical Receiver

The number of meshing elements for meshed model of receiver geometry is given Table.1

Table 1: Number of Meshing Element

Receiver Type	Number of Meshing Elements	
	Straight Position	Tilt Position
Cylindrical	2,77,897	2,83,246
Conical	4,35,965	3,69,319

3.2 Solver

In analysis, solution was derived using Pressure Implicit with Split Operator (PISO) scheme of fluent software. PISO is a pressure-velocity calculation procedure for the Navier-Stokes equations adapted successfully to steady-state problems. PISO involves one predictor step and two corrector steps and is designed to satisfy mass conservation using predictor-corrector steps as in [8]. Concentrated solar radiation is incident on the front side of both receivers. Adiabatic wall condition is assumed for insulated side. Incident solar radiation (W/m^2) is converted into the heat input (kJ).This heat input is taken as boundary condition for energy input. Average value of heat transfer coefficient for air surrounding the receiver is also given as input boundary condition to take into account, an effect of convective loss on the thermal efficiency. Inlet water temperature is taken 25°C for normal conditions and 50°C pre heating of water condition. All simulations are performed at 1 bar steam pressure (gauge) and 500 K focal temperature based on experimental data.

Table .2: Boundary conditions.

Component /Face	Initial Value/Boundary Condition
Front Surface	500 K
Inlet Water (With Initial Heating)	323 K
Inlet Water	298 K
Outside Air	310 K
Convective Air Heat Transfer Coefficient	5 W/m^{2}K

The pressure based solver technique and Transient solution method is used for simulation. In this study inner water flow is laminar and multiphase model is used for inner fluid condition. Water is considered as primary phase and vapour is considered as secondary phase. The solutions for the problem are obtained by solving the continuity equation, the momentum equations and energy equations simultaneously. (MUSCL) scheme used for solution of energy equation. Compared to the second-order upwind scheme, the third-order MUSCL has a potential to improve spatial accuracy for all types of meshes by reducing numerical diffusion, most significantly for complex three-dimensional flows, and it is available for all transport equations. For Transient formulation first order implicit method with time step size 0.1 second is used. In each time step, 20 iteration are performed.

3.3. Post-Processing

In post processing, the output of simulations from solver were directly converted to graphic form. In each case of experimentation namely - General condition, Pre heated water , Receiver tilted case, Simulations are converted into graphics for Water condition and temperature distribution on the surface of receiver. The volume fraction of water gave the amount of steam generated and the rate. Mass flow rate of steam is calculated by water evaporated divided by time it takes till achieving 1 bar pressure. These visual outputs were resultant of 20,000 iterations.

4. Result and Discussions

4.1 Results of Experimental Investigation

After reaching 1bar steam pressure at steady state condition, the Receiver water was decanted to collect unconverted water volume to compute steam generation rate. The results are tabulated along with Numerical results.

4.2 Results of Numerical investigation using CFD

4.2.1. Cylindrical Receiver

The initial water quantity 4000 ml which is half of the total volume of the receiver is as shown in Figure 5 by red colour.

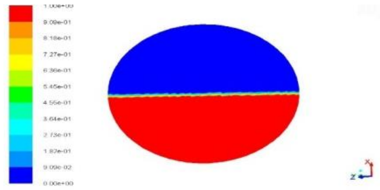
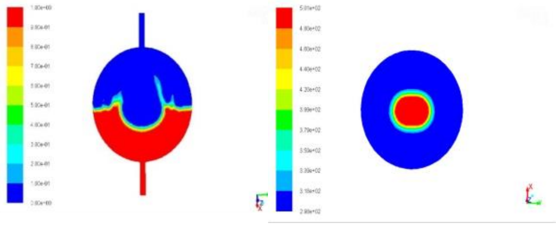


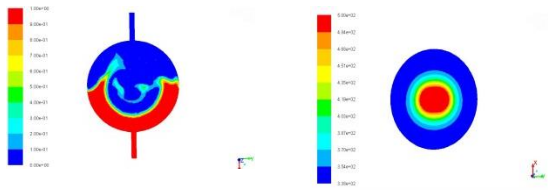
Fig 5 showing initial quantity of water in cylindrical receiver



(a)Volume of Water (b)Temperture Distribution

Fig .6 Simulated System for General Case(input Water at 25^{o}C) in Cylindrical Receiver

Fig.6 (b) shows temperature distribution on the outer surface of cylindrical receiver for General condition with a small temperature region of 500 K at focus to 359 K. The effect of this region shown in Fig. 6(a), is 360 ml water gets converted into steam till reaching 1bar pressure.



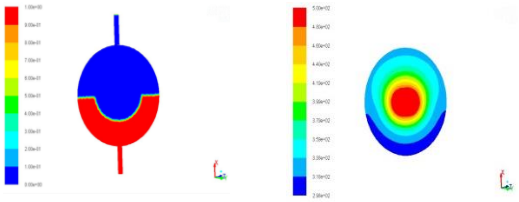
(a)Volume of Water (b)Temperture Distribution

Fig.7. Simulated System for Initial Heating Case(Water at 50^{o}C) in Cylindrical Receiver

In case of pre heating of water, receiver showed better performance than the general condition. Fig 7(b) shows that more heated surface area of receiver with temperature region varying from 500 K at focus to 370 K .The water quantity which converted into steam was 440 ml. Initial heating of inlet water enabled to reduce the time required for conversion, so the steam conversion is improved as visible in fig. 7(a)

Fig 8(b) shows that more heated surface region occurs in vapour region so it reduces the rate of conversion of water into steam due to lower contact of water and heated surface.

The temperature region varies from 500 K to 318 K, as shown in fig.8 (a).



(a)Volume of Water (b)Temperture Distribution

Fig 8. Simulated System for Tilting Case (Receiver at 45° to horizontal) in Cylindrical Receiver

4.2.2. Conical Receiver

Initial condition of water level into the conical receiver is as shown in Fig .9,which shows conical receiver with half filled water, fig.10 (a) and fig.11(a) shows volume of water after simulation.

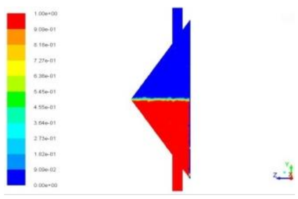
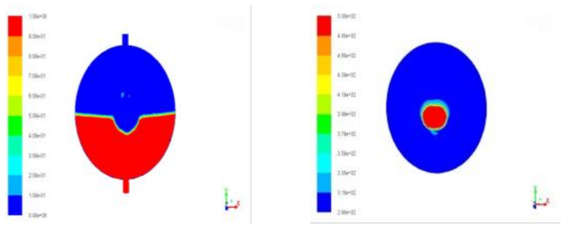


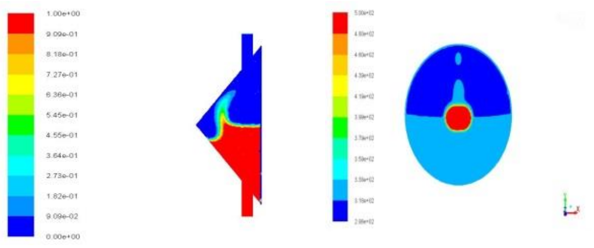
Fig 9 .Water Level at Initial condition in Conical Receiver

For general and initial heating condition small temperature region near tip of cone available for heating of inside water as shown in Fig.10 (b) and Fig.11 (b).This temperature region varies from 500 K at focus to 338 K. The quantity of water gets convert into steam is 320 and 325 ml.



(a)Volume of Water (b)Temperture Distribution

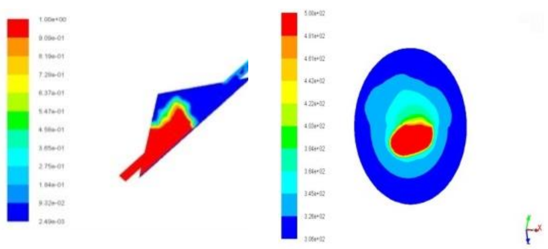
Fig 10. Simulated System for General condition(Water at 25°C) in Conical Receiver



(a)Volume of Water (b) Temperature Distribution

Fig 11 Simulated System for Initial Heating Case(Water at 50°C) in Conical Receiver

Fig .12(a) shows volume of water and Fig. 12(b) surface temperature distribution for conical receiver for tilting condition. Conical receiver attains more focus area than the other operating condition. In this case useful temperature region varies from temperature 500 K to 345 K, and covers more surface area than the other condition.360 ml water quantity converted into steam for this case. With tilt inside water quantity in contact with the focus region increases and as a result steam generation rate improves.



(a)Volume of Water (b)Temperature Distribution

Fig 12 Simulated System for Tilting Condition (Receiver at 45° to horizontal) in Conical Receiver

4.3 Validation of Numerical Results

Table 3: Comparison of Numerical and Experimental Mass Flow Rate for Cylindrical Receiver

Parameter	Mass Flow Rate Kg/hr Steam(kg/hr)		Percentage difference (%)
	Numerical	Experimental	
General Condition	0.95	0.89	6
Initial Heating	1.76	1.60	9
Tilt Angle	1.28	1.20	6.4

Table 4. Comparison of Numerical and Experimental Mass Flow Rate for Conical Receiver

Parameter	Mass Flow Rate of Steam(kg/hr)		Percentage Variation (%)
	Numerical	Experimental	
General Condition	0.64	0.60	6.4
Initial Heating	1.63	1.52	6.9
Tilt Angle	1.44	1.30	10.21

The numerical result shows good agreement between experimental and numerical values with maximum 10.21% variation. Higher quantity of water converts into steam in initial heating case and tilting case for cylindrical and conical receiver respectively.

The temperature distribution model using CFD in each case clearly makes it possible to visualize the steam generation and with changes in operating conditions associated changes in steam generation rate. Also it shows various temperature regions in the focal area changing with Changes in Operating conditions. The cylindrical receiver shows better steam generation rate as compared to conical in first two conditions as temperature region on the surface of the receiver is larger. For Conical receiver Tilt increases the temperature region and hence shows the best steam generation rate. In both receivers with initial heating the steam mass flow rate shows increase due to less thermal inertia.

5. Conclusion

Numerical and Experimental investigation was carried out for two types of receivers under three conditions of operation. The emphasis of the work was on investigating temperature distribution and mass flow rate of steam which is key point from further application design while in most of the past reference work such technique has been used to estimate and predict convective losses from cavity Receiver.

The investigation has showed that Cylindrical receiver gives better Steam generation rate than conical under general condition of operation since it has higher focus area that is concentrated area of solar rays . This mass flow rate of steam further improves with initial heating of input water as it helps to reduce thermal inertia. However tilting of receiver reduces the rate in cylindrical receiver as focus area starts reducing. The best of the result is given by Conical receiver with tilt . This has enabled to locate most optimum condition as Conical receiver with tilt to give the best of mass flow rate of steam.

The experimental results collaborate with Numerical investigation within 10% of each other thus validating the Numerical analysis using CFD. The visual output of temperature distribution helps to understand the change in focal area under each change of operation. It is expected now that the CFD model developed herein will be utilized for further investigative work on efficiency improvement.

Nomenclature	
D_{cyl}	Diameter of Cylindrical Receiver (mm)
D_{con}	Diameter of Conical Receiver (mm)
$(Vr)_{cyl}$	Volume of Cylindrical Receiver (litre)
$(Vr)_{con}$	Volume of Conical Receiver (litre)
η	Thermal Efficiency (%)
A_p	Aperature Area of Scheffler Reflector (m^2)
M_s	Mass Flow Rate of Steam (kg/hr)
I_{bn}	Direct Solar Radiation(W/m^2)
I_D	Diffused Solar Radiation(W/m^2)
I_G	Global Solar Radiation(W/m^2)
CFD	Computational Fluid Dynamics
PISO	Presure Implicit with Split Operator

References

[1] M. Prakash, S.B. Kedare, J.K. Nayak," Investigations on heat losses from a solar Cavity Receiver", Solar Energy,vol 83 , pp.157–170,2008.

[2] Hao Li, Li Rong and Guoqiang Zhang , "Study on convective heat transfer from pig models by CFD in a virtual wind tunnel", Journal of Computers and Electronics in Agriculture, vol 123, pp. 203-210, 2016.

[3] Gawali B.S., Kawade A.G., Deshpande G.S, "Thermal analysis of Scheffler cavity receiver for distribution and heat losses during solar period", ISHMT-ASME Heat and Mass Transfer Conference ,Mumbai,India pp. 1-6,4-6January , 2010.

[4] M. Prakash, "Numerical Study of Natural Convection Heat Loss from Cylindrical Solar Cavity Receivers", ISRN Renewable Energy, Vol 2014, pp. 1-7,Article ID 104686, 2014.

[5] N. Sri Gokilavani, "Experimental Studies and CFD Modelling on Solar Distillation System", International Journal of Innovative Research in Science Engineering and Technology, Vol. 3, Issue 9, pp.15818-15822, September 2014.

[6] Ashish Kumar, "Experimental and Simulation Study on Natural convection Solar Food Dryer", International Journal Of Science, Engineering And Technology, ISSN: 2348-4098 Volume 02 Issue 06, pp. 987-996,July 2014.

[7] John D.Anderson,Computational Fluid Dynamics, McGrawHill,1995,pp.60-66.

[8] H.K.Versteeg and W.Malalasekera, "An Introduction to Computational Fluid Dynamics(The Finite Volume Method",2ndEdition,Pearson Education Limited,2007,pp.150-154.

[9] Grigorios Iordanou, "Experimental studies coupled by Computational Fluid Dynamic (CFD)findings for the of Heat Transfer enhancement in Flat Plate Solar Water Collectors",IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE),Vol 11, Issue 3 PP. 50-57, May- June.2014.

[10] Mohamed B. Gadi, " Design and simulation of a new energy-conscious system (CFD and solar simulation)", Applied Energy,vol 65 ,pp. 251-256,2000.

[11] R. D. Jilte, S. B. Kedare, and J. K. Nayak, "Comparison of Cavity Receivers with and without Mouth-Blockage of Different Shapes and Sizes Used in Paraboloid Dish Applications", Journal of Fundamentals of Renewable Energy and Applications,Vol. 2 Article ID R120306, pp. 1-6 ,2012.

[12] Ketan Ajay, Lal Kundan" Experimental and CFD investigation on the efficiency of Parabolic solar collector involving Al_2O_3/H_2O (DI) Nano fluid as a working fluid", International Journal of Renewable Energy Research,vol6, no2, pp.399-401,2016.

[13] Rohan Bajaj, Amrit Raj,T Srinivas "Mathematical modelling and Analysis of variation in Surface of solar Scheffler concentrator for seasonal variations" Energy Efficient technologies for Sustainability (ICEEETS), Nagercoil,India, 78-83, 7-8 April 2016

[14] Lan Xiao, Shung Ying,Wu, You Rong Li" Effect of cavity wall temperature and opening ratio on the natural convection Heat loss characteristics of a solar cavity receiver" Computer Distributed control and intelligent Environmental monitoring (CDCIEM), Changsha, china,1112-1115, 19-20 Feb 20111

[15] Hu Zhuohuan, Zhang Leyi,Yang Ho " Analysis of compound Parabolic collector with different cavity structures" Information Teachnology, Net working, electronics and Automation control conference IEEE,Chonqing,China pp 464-468, 20-22 May 2016

[16] Jianbin Fang, JinJia Wei, XunWei Dong "A calculation Method for evaluating Thermal loss for cavity receiver" Power and Energy Engineering conference, Wuhan, China, 27-31 March 2009.

[17] Sarihassoun Zakaria,Aliune Khaled,Henaoui Mustapha "Experimental study of a flat plate solar collector equipped with concentrators", International Journal of Renewable Energy Research, vol 7, no3, pp.1028-1031,2017.

[18] MOSBAH charaf Abdelkarim,TAD JINE Mohmed, CHAKIR Messaond, BOUCHERIT Mohmad seghir "On the control of parabolic solar concentrator : The zipper approach" International Journal of Renewable Energy Research, vol 6, no3, pp.1101-1108, 2016.

[19] Dr Ankur Geete,Somesh Kothari,Pratik Likhar,Anil Singh,Anki Saini,Rahul Sahu " Experimental Analysis on fabricated parabolic solar concentrator with various flowing fluids" International Journal of Renewable Energy Research, vol 6, no4, pp.1454-1463,2016.

[20] Mohamed H. Ahmed,Mohamed Rady, Amr M A Amin,Fabio Maria Montagnino, Filippo Pardes "Comparision of thermal and optical performance of Linear Fresnel and parabolic trough concentrator" International conference on Renewable Energy Research and Application ICRERA,Palermo,Italy,pp 626-629,22-25Nov 2015.

[21] Sameer Simms, Jean –Francois Dorville "Thermal performance of a hybrid photovoltaic –Thermal collector with a modified absorber" International conference on Renewable Energy Research and Application ICRERA,Palermo,Italy,pp 600-605, 22-25Nov 2015.

[22] Adan Al-Ghasem, Ghassan Tashtoush, Mohammed Aladeemy, "Experimental study of tracking 2-D compound parabolic concentrator(CPC) with flat plate

absorber" International conference on Renewable Energy Research and Application ICRERA, Madrid, Spain, pp 779-782, 20-23 Oct 2013.

[23] Wahiba Yaici,Evgueniy Entchev "Prediction of the performance of a solar energy system using neuro-Fuzzy interface system", International conference on Renewable Energy Research and Application ICRERA, Milwaukee,WI,USA pp 601-604, 19-22 Oct 2014.

[24] Evren M Toygar,Alpar Yazar, Tufan Byram,Mustafa Tustan, Omar Faruk Kaya, Ouzhan Das Huseyin Calmaz "Design and development of solar flat mirror and heat storage System" , International conference on Renewable Energy Research and Application ICRERA, Milwaukee,WI,USA pp 821-827, 19-22 Oct 2014.