Advances in Engineering and Technology
An International Journal

Computational Fluid Dynamics (CFD) Modelling & Simulation of 3D-Single Slope Basin Type Passive Solar Still to Improve Distillate Yield

Shah Nawaz Ansari [1,2*], Nawraj Bhattarai [1*], Sanjaya Neupane [1], Iswor Bajracharya [2]

[1] Department of Mechanical and Aerospace Engineering, Pulchowk Campus, IOE, TU, Nepal

[2] Faculty of Technology, Nepal Academy of Science and Technology (NAST), Nepal

Emails:
ansari.nawaz1406@gmail.com (S.N. Ansari),
bnawraj@ioe.edu.np (N. Bhattarai),
sanjaya@pcampus.edu.np (S. Neupane)
iswor1@yahoo.com (I. Bajracharya)
*Correspondence to: N. Bhattarai and S.N. Ansari

Abstract

The Industrialization and population growth are causing a rapid surge in water demand. Water pollution and industrial waste mixing with water make the freshwater impure. Therefore, the need for pure water is increasing. One of the solutions to this in rural areas could be using solar still as a distillation technique using solar energy. Despite the many advantages of being environment-friendly, simple, easily affordable, and abundant solar energy, low productivity is a major disadvantage of solar still. While adding components like concentrators and collectors with it can boost output but it also increases costs. Thus, the primary challenge is to enhance the distillate yield i.e. productivity without significant increase in cost. Investigating and optimizing various performance parameters of solar still can improve the productivity of solar still. Hence, this paper performs the computational fluid dynamics (CFD) analysis to identify the parameters and their behaviors. FLUENT module in ANSYS is used for modelling the condensation and evaporation phenomenon of solar still. Simulation for different water volumes in basin i.e. 20 litres, 15 litres and 10 litres was done. Result showed 7% increment in productivity when volume was reduced from 20 litres to 10 litres. Finally, necessary conclusions and recommendations were given.

Keywords: *ANSYS; Clean Energy; Desalination; Modelling; Simulation; Solar Still; Productivity.*

1. Introduction

Along with food and air, water is also a primary basic need in people's lives. Water is responsible for the development and welfare of any society. Each aspect of human life is directly or indirectly linked to the availability of clean drinking water. Hence, the availability of clean and pure drinking water is one of the world's biggest issues in the present context [1].

According to Deshmukh and Thombre (2017), various parameters, such as operational, design and climatic parameters, on has impact on the solar still's performance which is based on various numerical and experimental research. The essential parameters evaluated were solar radiation, a tilt angle of the glass, depth of saline water, and blackened substance. [2].

The water desalination method is a traditional and effective method to increase the supply of pure potable water as per the requirements by converting saline water into clean water. Access to clean water causes positive benefits such as good health, hygiene, improvement in standard of living and quality of life, and so on [3].

Solar still is a device that distils or desalinates water with the help of solar energy through the evaporation and condensation principle [4]. Commercial distillation processes such as Reverse Osmosis (RO) technology, etc., are not feasible in remote areas due to high fossil fuel costs and lack of electricity. Due to this, solar still is emerging as potential solution to fulfil potable water requirements in the remote or rural areas due to its beauty of being cheap, abundant and pollution free [5].

The main drawback of solar still is its low production of distillate water. Hence, the greatest challenge associated with this is to improve distillate water production without increasing its cost. This can be resolved by optimising the crucial parameters of solar stills. CFD analysis can help investigate the significant parameters associated with solar stills. CFD analysis is a suitable option since it reduces the physical effort and time required for experimentation. [6]. Integrating other auxiliary devices such as concentrators, solar ponds, and collectors with the solar still could enhance distillate water production. However, it will substantially drive up the cost of solar still. The drawback of low distillate production could be improved if the parameter is investigated. After investigating the parameters, optimizing them could significantly overcome this challenge. Thus, the current paper performs CFD analysis to investigate the performance parameters, which can be later optimized for improving the productivity of solar still.

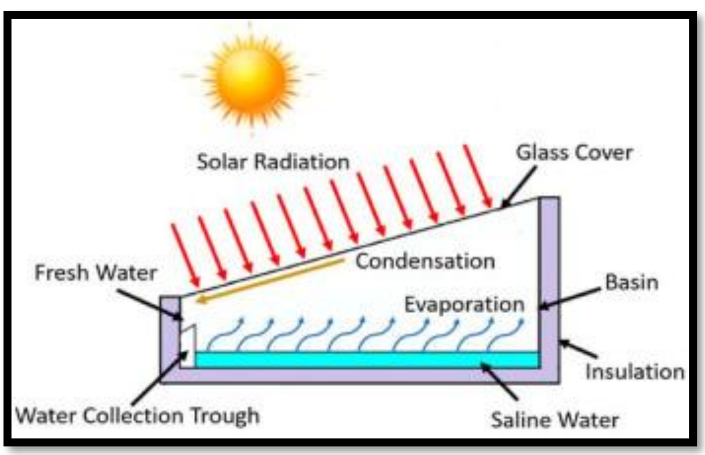

Figure 1: Solar still and its phenomenon [7]

Figure 1 shows the working phenomenon and main component of solar still. The basin is generally made up of Galvanised Iron (GI) sheet, and the top cover is made of transparent glass or plastic. The inclination of glass cover is according to the latitude angle of particular location. The working mechanism of solar still is the same as that of the hydrological cycle of nature (Kabeel & Abdelgaied, 2016). Solar radiation coming from the sun strikes on the glass cover and goes to the inner surface of basin causing vaporisation of water. Vapour goes up and leaves salt and other heavy impurities behind. After that, condensation takes place, resulting in water being collected after flowing from the collecting channel with the impact of gravity (Alwan, Shcheklein, & Ali, 2019).

It was found that the major research gap is identification of performance parameter of solar still and optimize it to improve the productivity. Thus, the main objective of this research is to perform CFD analysis of 3D-single slope basin type passive solar still. As a result of CFD analysis, key parameters affecting the productivity will be identified. Also, the impact of volume of water inside the basin will be studied by simulation.

2. Research Methodology

The existing problems of low production was found out with the help of literature review. Modelling and simulation software was used for CFD analysis in the current research. The methodology used for CFD analysis can be understood with the help of a flowchart, as shown in Figure 2.

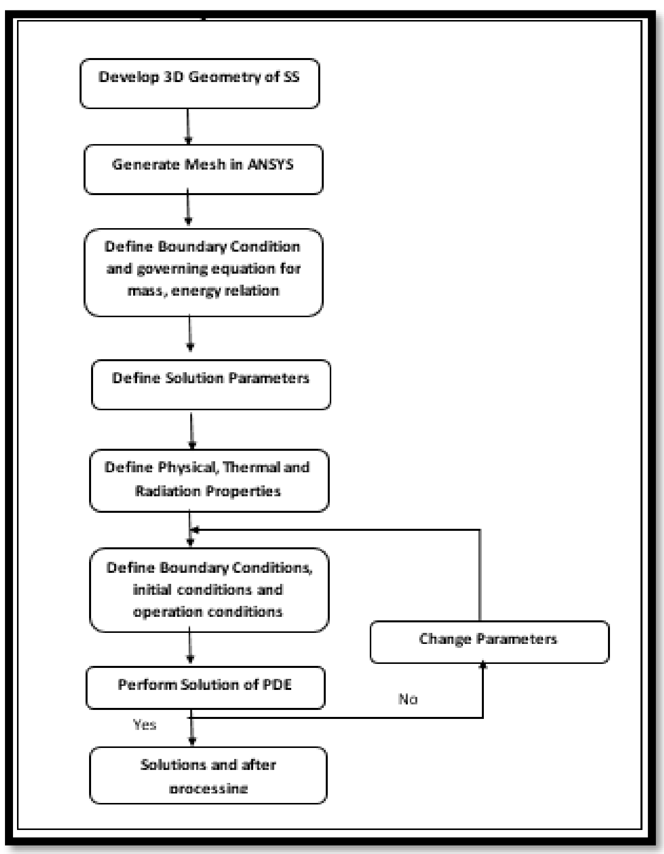

Figure 2: Flow chart of research methodology

Initially 3D geometry of solar still was developed with the help of SpaceClaim with proper dimensions taken as per standard physical model. Meshing was done after that with proper size and type. Various equation was applied by the simulation software by dividing the geometry into large number of tiny elements of specified type. Once meshing was done, different boundary conditions and governing mass, energy and momentum relations along with various input parameters for solution was defined. Similarly, different physical, thermal and radiation properties was also defined where required. After that, the result was obtained by performing simulation on software. Finally, the result obtained from the simulation of CFD are interpreted to know the key parameters.

2.1 Solar Still Modelling and Simulation

Geometric model of the solar still with required necessary specifications of design as per physical model was developed. The present research performs CFD analysis using ANSYS Workbench software. The basin area is considered as 0.8 m X 1.0 m and thickness of glass is 5 mm. Angle of inclination of condensing glass is taken as the latitude angle of Kathmandu i.e. approximately 27 degrees. Figure 3 shows the detailed drawing of the solar still made by SpaceClaim Module in ANSYS [8].

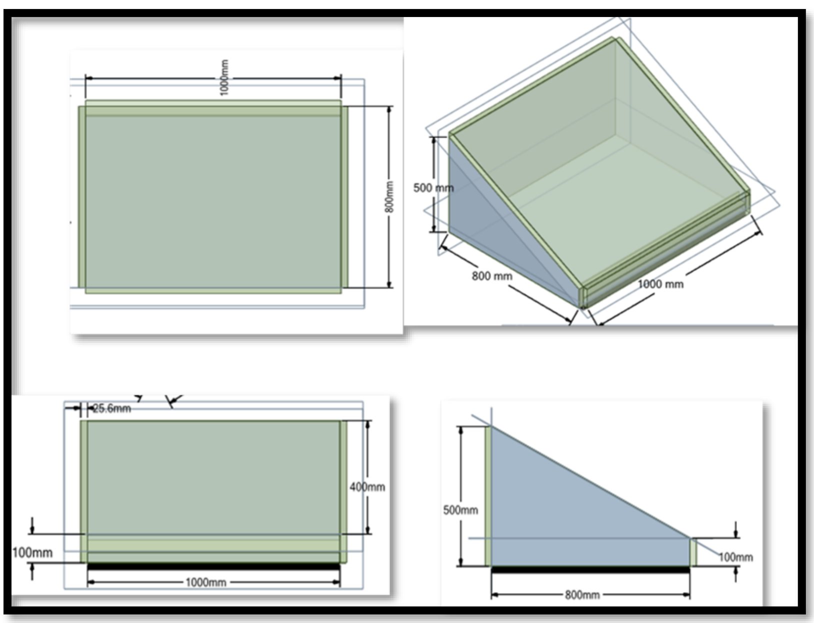

Figure 3: Dimension of Solar Still Drawn on SpaceClaim in ANSYS [8]

The DesignModeler module of ANSYS Workbench was used to develop the 3D model for meshing as per the model's dimensions in SpaceClaim. The basin was kept black for enhanced radiation and absorptivity rectangular basin which is assumed to be made up of GI sheet.

Meshing of the 3D modelled solar still was done after geometric modelling as shown in Figure 4. Hexahedral meshing was selected in the present research as there is no curved or sophisticated surface in the current design of the solar still. The element size and maximum size were both set to 0.013 m. Smoothing was kept on high. Linear element order was taken. Once mesh was generated, the overall number of nodes is 123240, and the total number of components in the created mesh is 114345, which is sufficient for the current complexity of the solar still problem. The mesh quality is also assessed using skewness and aspect ratio.

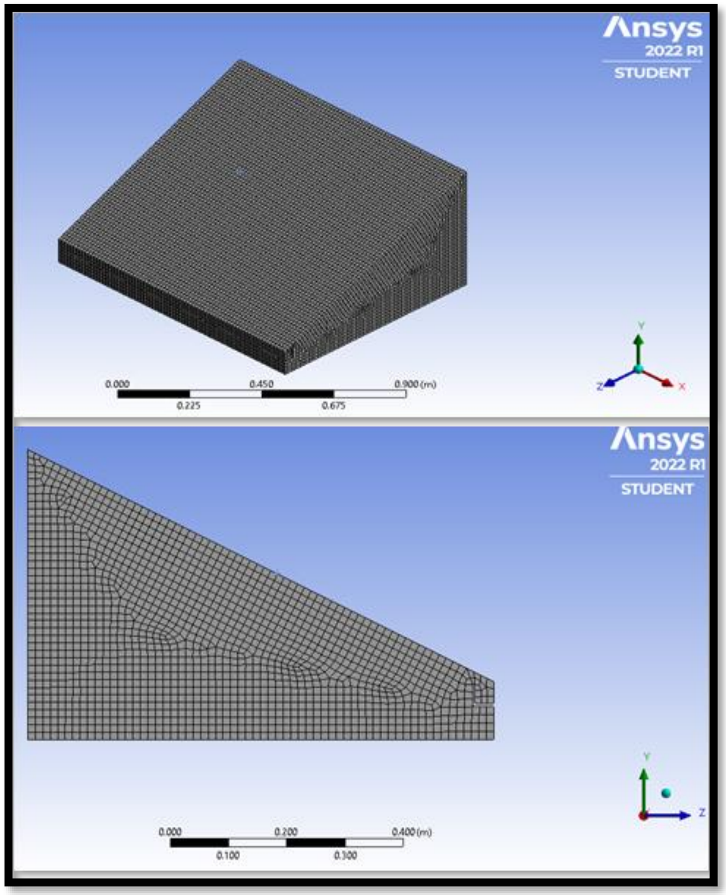

Figure 4: Modelling and Meshing of Solar Still

FLUENT is a very useful module or tool in ANSYS that permits the transmission of heat and mass to the model's defined boundary and region. The application of FLUENT for fluid flow embedded in ANSYS allows for the easy simulation of a variety of problems involving convection, radiation, and mixed flows [9]. Current analysis focuses on the simulation of evaporation and condensation mechanism of water inside the solar still through CFD analysis. Various assumptions have been made for the present research

2.2 Assumptions made during Simulation of Solar Still

During the simulation of a three-dimensional multiphase single slope basin type passive solar still, the following assumptions were considered. To simplify the analysis, some criteria were assumed which are as follows [10].

Solar still was assumed to have adiabatic walls since real model is provided with insulation.

Constant physical and thermal properties of materials such as specific heats, thermal conductivity, and density was assumed with negligible inlet velocity.

Basin and Glass temperature was assumed to be homogeneous and free convection is taken into consideration.

2.3 Input Parameters and Solution Method Used for Modelling in ANSYS

To turn a physical problem into a virtual model in FLUENT, appropriate parameters must be modeled for simulation. Various input parameter include material used, type of phases, cell zone, operation condition, type of models, etc. for the simulation. On the other hand, parameters for solution control and methods include type of pressure-velocity coupling, spatial discretization method and solution controls.

The various input parameters and parameters for method and solution control of single slope solar still in ANSYS FLUENT is shown in the Table 1.

Table 1: Input Solve Parameters and Solution Method Control Parameters in ANSYS FLUENT

Input Solver Parameters in ANSYS FLUENT	
1. Solver	• Space: Three dimensional Space used • Time: Transient condition, 1st order, implicit function • Viscous model: k-epsilon (2 equation) with standard wall functions • Multi-phase model: Mixture model • Radiation model: Rosseland model with solar tracking system and solar loading as per the value of Kathmandu
2. Material Used	• Solid Material: Galvanized iron sheets (GI), Aluminium and Glass • Fluid Material: Air, Water-liquid, Water-vapor
3. Phases	• Number of phases: 3 • Phases involved: air, water-liquid, water-vapor • Primary phase: air • Secondary phase: water-liquid and water-vapor
4. Cell Zone	• Source term – Energy source
5. Operating condition	• Gravity axis: Y-Direction • Gravity value taken: -9.81 m/s2 • Operating pressure and temperature: 1.01 bar and 288.16K
Parameters for method and control of solution of solar still in ANSYS FLUENT	
1. Pressue-velocity coupling	• Simple
2. Spatial Discretization	• Pressure: PRESTO • Momenturm, Energy, Liquid Volume Fraction: 1st order upwind
3. Solution Controls	• Governing equations: Momentum, Continuity, Energy, Std. Initialization Method.

3. Results and Discussion

Using various input parameters and values, CFD modeling and simulation have been performed for 3D-single slope passive solar still. Following the initialization of the solution, various contour types were produced as the outcomes. The temperature of the condensing glass, the temperature inside the solar still where the phenomenon occurs, and the temperature of the water inside the basin are the main parameters that affect how well the solar still operates. It follows that the temperature gradient between the water in the basin and the condensing glass determines the solar still's output productivity. The simulation has produced contours that depict the temperature profile. To properly reflect the contours, the temperature range was between 303 K and 343 K. Following information could be retrieved from the contour of temperature profile.

i. Water begins to evaporate and turn into vapor when solar energy from the sun strikes glass due to which the inside temperature of the solar still rises with time.

ii. As solar radiation increases, the interior temperature continues to rise until it reaches its maximum and afterwards it begins to fall. The behavior of the temperature is comparable to that of solar radiation.

iii. Furthermore, the contour of the solar still shows that the temperature is proportional to the solar radiation as a result of which the inside vapor temperature of the solar still also keeps on rising with glass and water. On the Other hand, the contour also shows that the temperature profile of the condensing glass is also rising with time.

iv. The trend for glass temperature is comparable to that of water temperature and sun radiation. Condensation of the vapor occurs because of the difference in temperature between temperature of the glass and the temperature of the evaporated water.

Figure 5 displays a snapshot showing water temperature contour collected at during 12:00 to 14:00. In a similar way, Figure 6's depiction of the glass temperature profile shows during 12:00 to 14:00 at which the solar radiation as well as basin temperature is high. The image depicting the contours of condensing glass makes it evident that the temperature is higher near the upper edge of the glass and lower toward the bottom end. The distilled water in the condensing glass flows from the higher end to the lower, resulting in the reduced temperature at the lower end.

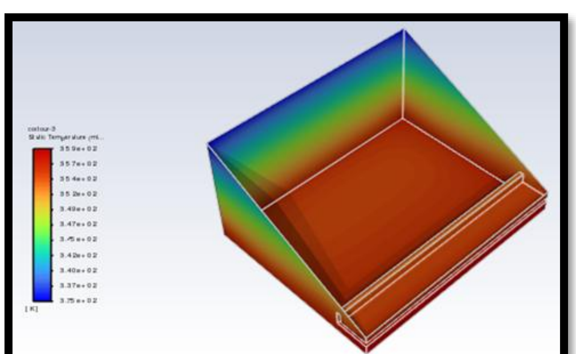

Figure 5: Temperature profile of water in basin

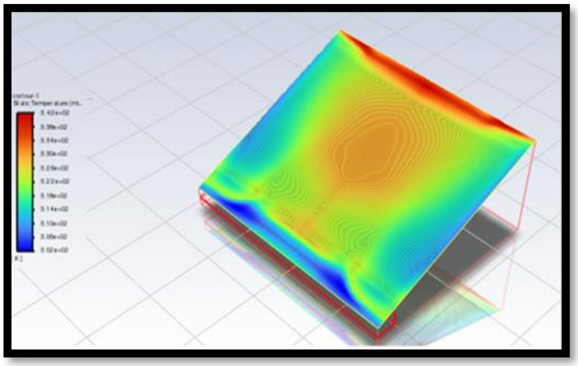

Figure 6: Temperature Profile of Glass

Because the water and absorber experience the most heat and evaporation, respectively. As seen in the Figure 7, the temperature at the basin/ absorber is significantly higher than the glass and inlet vapor.

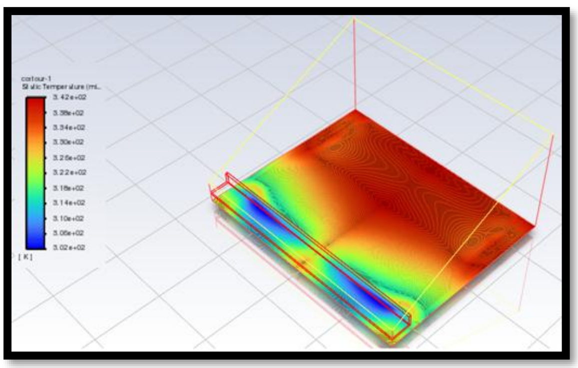

Figure 7: Temperature Profile of absorber

The contour of the vapour volume fraction inside the solar still at day's peak radiation is depicted in Figure 8. It demonstrates that the solar still's upper surface, where water condensation occurs, is where the largest volume of vapor is found.

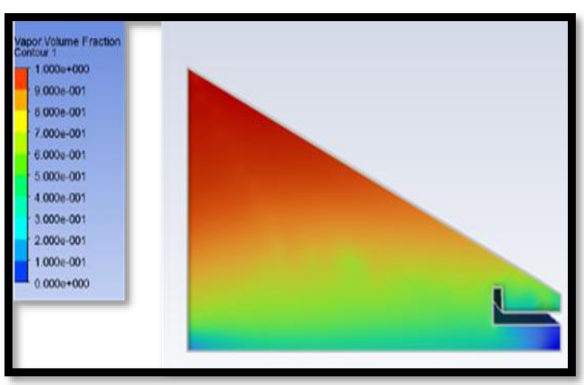

Figure 8: Volume fraction of vapour

Thus, the major parameters which is responsible for the distillate yield in solar still are glass temperature, water temperature, solar radiation, etc. Other parameter which could be varied are thickness of glass, material of absorber, material of basin, insulation thickness etc.

Effect of Water Depth or Volume of Water on Productivity or Distillate Yield

Many vital variables are associated with the impact on the water quality obtained as distillate yield in solar still. Some of the variable are basin material, insulation thickness and type, temperature and volume of water or depth of water inside the basin. Since the depth of water directly affects the volume of water in the basin.

The CFD simulation is done for three volumes of water that is 20 litres, 15 litres and 10 litres respectively. Distillate output was calculated with the help of temperature of glass and water obtained from simulation result. The behavior of glass, water and yield is shown with the help of graph. Figure 9, Figure 10 and Figure 11 shows the solar still behavior at when initial water volume inside basin is kept 20 litres, 15 litres and 10 litres respectively.

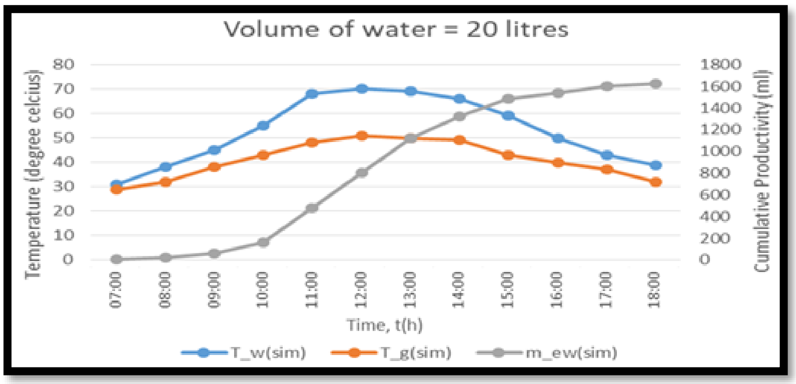

Figure 9: Variation of temperature and cumulative productivity (20 litre)

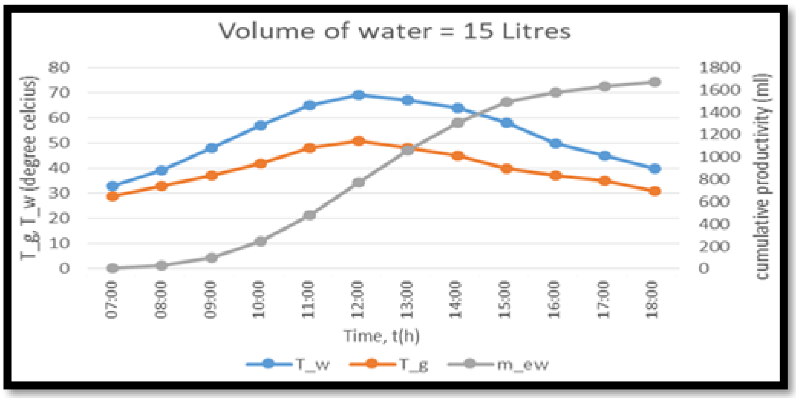

Figure 10: Variation of temperature and cumulative productivity (15 litre)

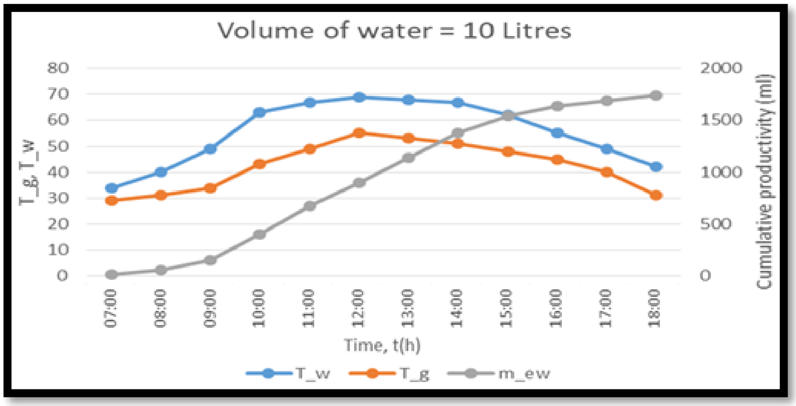

Figure 11: Variation of temperature and cumulative productivity (10 litre)

Thus, it is clear that volume plays key role in heating process of water inside basin. Temperature of basin rises at higher rate with low water volume due to increase in evaporation rate as a result of which production of distillate water increases. The distillate yield output was found to be 1620 ml, 1671 ml and 1738ml with respective water volume of 20 litres, 15 litres and 10 litres. Therefore, it was found that when the water volume was reduced from 20 litres to 10 litres, there was increase in productivity of distillate water by 7%. Thus, the volume is inversely related to productivity. Due to the fact that less water has a lower heat capacity and warms up more quickly, the output water rose as the volume decreased.

4. Conclusions

This research's main objective was to perform CFD modelling and simulation to investigate the key factors that may be optimized to increase distillate water productivity. Based on the simulation results, major parameters are found to be glass temperature, basin temperature, water temperature and solar radiation. Utilizing the ANSYS Workbench's design modeler module, the solar still's geometry was produced. By providing the correct input parameters to the solver in FLUENT in ANSYS, modeling and simulation were completed. Simulation was carried out for different volume of water inside basin i.e. 20L, 15 L and 10L respectively. It was found out that volume is inversely related to the distillate yield. It was found that there was 7% increment in distillate yield when the volume is reduced from 20 litres to 10 litres. The following recommendations for future works are made based on the knowledge and expertise obtained during the CFD analysis in current research. This will aid in improving the outcome and applying the current research to future studies:

- Future research may examine the simulation of a solar still with many layers of glass and an open type solar still system.
- Since the material of a basin directly affects its temperature, several basin materials can be used.

- CFD analysis could be done by varying the shape of solar still and comparing with the result of current research.
- When examining the solar still result, more precise radiation modeling, such as that which uses the DTRM or DO models, might be employed. The outcome could then be compared with the Rosseland model.

References

[1] M. A. Pakdel, M. Hedayatizadeh, S. M. Tabatabaei and N. Niknia, "An experimental study of a single-slope still with innovative side-troughs under natural circulation mode.," *Desalination,* vol. 422, pp. 174-181, 2017.

[2] H. S. Deshmukh and S. B. Thombre, "Solar distillation with single basin solar still using sensible heat storage materials.," *Desalination,* vol. 410, pp. 91-98, 2017.

[3] S. Zhou, L. Gong, X. Liu and S. Shen, "Mathematical modelling and performance analysis for multi-effect evaporation/ multi-effect evaporation with thermal vapor compression desalination system," *Applied thermal engineegin,* vol. 159, pp. 113-759, 2019.

[4] H. W. Zachritz, K. Stevens, L. Mimbela, R. Polka and L. Cisneros, "Distillate water quality of a single basin solar still: Laboratory and field studies," *Solar Energy,* vol. 76, pp. 635-645, 2004.

[5] S. B. and J. Patel, "Recent Developments in solar desalination with thermal energy storage," *International Journal of Engineering Research and Applications,* vol. 7, pp. 28-36, 2017.

[6] Ansys Inc., "Fluent, Ansys," in *Ansys Fluent Theory Guide*, USA, 2011.

[7] A. Johnson, L. Mu, Y. H. Park, D. J. Valles, H. Wang, P. Xu, K. Kota and S. Kuravi, "A Thermal Model for Predicting the Performance of a Solar Still with Fresnel Lens," *Water,* vol. 11, no. 9, p. 1860, 7 September 2019.

[8] S. N. Ansari, "Computational Fluid Dynamics Modelling and Experimental Verification to Enhance Productivity of Distillate Water in Three Dimensional Single Slope Basin Type Passive Solar Still," I.O.E. Pulchowk Campus, Lalitpur, 2022.

[9] A. Fluent, Ansys Fluent Theory Guide, USA: ANSYS Inc., 2017.

[10] R. V. Dunkle, "Solar Water Distillation: The roof type still and a multiple effect diffusion still," in *International Heat Transfer Conference*, Boulder, 1961.