Journal of Mines, Metals and Fuels, *71(10): 1689-1696; 2023. DOI: 10.18311/jmmf/2023/35865*

Print ISSN : 0022-2755

Journal of Mines, Metals and Fuels

Contents available at: *www.informaticsjournals.com/index.php/jmmf*

CFD Analysis of Refrigerator Integrated with Atmospheric Water Generator

Aniket V. Deshmukh[1], Nititsh Kumar Gautam[2*] and Anoop D. Shirbhate[3]

[1]Research Scholar, Department of Mechanical Engineering, SJJT University, Jhunjhunu - 333010, Rajasthan, India; aniket.deshmukh111292@gmail.com

[2]Assistant Professor, Department of Mechanical Engineering, SJJT University, Jhunjhunu - 333010, Rajasthan, India; nitish08m50@gmail.com

[3]Associate Professor, Department of Mechanical Engineering, Prof. Ram Meghe Institute of Technology and Research, Badnera - 444701, Maharashtra, India; adshirbhate@gmail.com

Abstract

This Research article discuss about the study of performance analysis. This system is a combination of refrigerator and Atmospheric Water Generator (AWG). The purpose behind integrating two devices is to get benefit of both products together, refrigerator will act as a domestic refrigerator plus the atmospheric water system will generate the fresh distilled water to drink. This system is designed to generate a water of 20 liters per day. Also, it is designed in such a way that, this AWG can attach with various capacity is the refrigerator. Performance evaluation of this system is done with the help of computational fluid dynamics. Fluent module is used in ANSYS software. Various levels of parameter are evaluated, variables considered in the study were Relative humidity, velocity of air in AWG heat exchanger, and temperature of air. The output parameters evaluated like water generation capacity of AWG, Coefficient of Performance (COP) of AWG and refrigerator and heat exchanger efficiency. The results of CFD shows positive results as per design. 24th case has given highest COP 1.26. General, range of COP lie between 1.127 to 1.260. COP of Standard Refrigerator remain between 1.437 to 1.593. Also, it means COP of other two cases are more. As far as COP of Refrigerator concern, COP with AWG is heist i.e. 1.91. But it cannot be clearly state that, COP with AWG is more than COP without AWG.

Keywords: *Atmospheric Water Generator, Performance Analysis, Refrigerator Integrated with AWG, CFD*

1.0 Introduction

Purpose behind the use of CFD in this research study is to check a design of the AWG. The system was designed and then modelled in CREO. The geometry created was used in ANSYS fluent to validate the design. Also, the AWG is combined with the Refrigerator. It means condenser of refrigerator is fitted in the body of AWG. In AWG, there are one evaporator coil and two condenser coils are present. CFD may help for research study, by validating heat exchanger design. It may predict the results that someone may gate from the actual model. If there is any problem with design, researcher can change or modify the design. It will save time and cost of fabrication and modification[1].

Another purpose of use of CFD is to create an exact and same virtual experimental setup in comparison with actual working model of actual experimental setup. As there is one of the best ways of study is experimental investigation, but this method also needs validation or verification. Therefore, CFD is used in this study[2].

*Author for correspondence

2.0 Literature Review

Using two-dimensional and three-dimensional Computational Fluid Dynamics (CFD), the flow of milk through a system comprised of two corrugated plates was analyzed[3]. Two-dimensional calculations can indicate the effects of the corrugation form, but three-dimensional calculations are required to ascertain the weight contribution of the corrugation orientation. The only waves affected by the movement of incoming water are the first three. The developed model may provide a positive qualitative validation for the simulation's results. Calculations may be performed to determine the potential locations of hotter zones created by turbulent backflows adjacent to the wall. Because these are the areas most susceptible to fouling, it is essential that they be situated as far from the remainder of the ship as is practically possible[4]. Consequently, Computational Fluid Dynamics (CFD) may be considered a valuable asset for enhancing the efficacy of the design of plate heat exchangers[5].

Beginning in utilizing heat transfer enhancement technologies can improve the performance of heat exchangers in order to accomplish a particular heat transfer mission. These strategies can be broadly classified as either aggressive or passive. Active procedures necessitate the application of external forces, such as an electric field, sound waves, and surface vibrations, among others. For passive techniques to be effective, particular surface geometries or fluid additives are required. The use of curved tubes in a variety of heat transfer applications is one of the passive heat transfer enhancement methods that has found widespread use[6]. This is due to the fact that curved pipelines enable for more efficient heat transfer. This article summarizes the research conducted on heat transfer as well as the characteristics of single-phase and two-phase flow in curved pipelines[7]. This article will cover the following three basic classifications of curved tubes: helical coiled tubes, spiral coiled tubes, and other coiled tubes. This article provides an analysis of the established relationships between friction factors between single-phase and two-phase fluxes and single-phase heat transfer coefficients[8].

This investigation required the development of a digital model of a four-channel plate heat exchanger using computational fluid dynamics. Using Three-Dimensional Computational Fluid Dynamics (3D CFD) and a one-dimensional plug-flow model (1D plug-flow model), heat burden estimates were developed and compared to the actual results. In this study, both parallel and series flow topologies were investigated[9]. The channels, plates, and conduits of the exchanger are modeled using CFD, and the model accounts for flow variations both within and across channels. The results of the CFD analysis and the experimental data correspond quite well, especially in terms of series structure. According to the findings of a study conducted by, the heat transfer characteristics of a double-pipe helical heat exchanger were numerically analyzed in order to determine the impact of fluid thermal properties on heat transfer. The first experiment employed three distinct Prandtl values (7.1, 12.6, and 69.3) and the second experiment employed temperature-dependent thermal conductivities. The assumption of a linear relationship between fluid temperature and thermal conductivities led to the formulation of the baseline assumptions[10]. Six unique fluid-dependency models were constructed. In the second experiment, we utilized a configuration with both parallel and counterflow. Initial research revealed that the inner Nusselt number was dependent on the Prandtl number, especially for low Dean numbers. This correlation may have resulted from the fact that different entry lengths are susceptible to hydrodynamic and thermal effects[11]. Correlations between the Nusselt number, the Prandtl number, and a modified Dean number are provided to illustrate how heat conduction occurs through the annulus. The findings of the second phase of the investigation revealed that when the Dean number was modified, the correlation between it and the Nusselt number was significantly enhanced[12]. In the counterflow design, heat transfer rates were substantially higher than in the parallel flow design, but the ratio between the dependent and independent thermal characteristics remained the same[13].

A simulation model for an independent atmospheric water generator powered by photovoltaic modules has been developed and presented. The generic model created can be applied to simulation and design tasks. The model needs as input the long-term average weather data, the technical details and performance metrics of the AWG, PV modules, batteries, and regulator[14]. The created model was put through a full year of hourly simulation. The system is capable of producing water even without the use of external electricity[15].

1 m/s in the axial direction of the fins was the inlet boundary condition. At the entrance, humid air moved at

a predetermined speed, and at the downstream boundary condition, the atmospheric pressure served as the outflow, where there is no pressure gradient[16]. Starting at a steady temperature, the air flow began. For each of the three different ambient conditions, three test scenarios were run. A simulation runs in 2780.266 s on an Intel i3 processor with 4GB of DDR3 RAM. Three different sets of numerical results for the same mesh were produced. Temperature, humidity distribution, and pressure change within the flow duct were all numerically calculated for the various test conditions[17].

3.0 Methodology

For this study many models are made and assembled. Models like, refrigerator condenser, AWG heat exchanger, various parts of AWG. Specially for the CFD purpose, models of fluid are made with the system component. Model to show flow of fluid like refrigerant flowing through the condenser tube, refrigerant flowing through the evaporator tube, air flowing around condenser tube, evaporator tube and from inside heat exchanger walls also. In the Figure 1a, 11 to 12 parts are assembled but, during study these models are used for analysis in various parts. Because the case of the condensation around evaporator section and case of conditions in condenser section is of heat rejection both are different.

Another purpose behind dividing the heat exchanger is to reduce complication related the Ansys software. Working in Ansys creates heavy files to handle by the computer system. Ansys needs heavy configuration computers to work. In general Laptop or Computer with 4GB graphics card, 16 GB RAM, CORE-i7 processor is ok to handle 2D analysis problem. Therefore, CAD

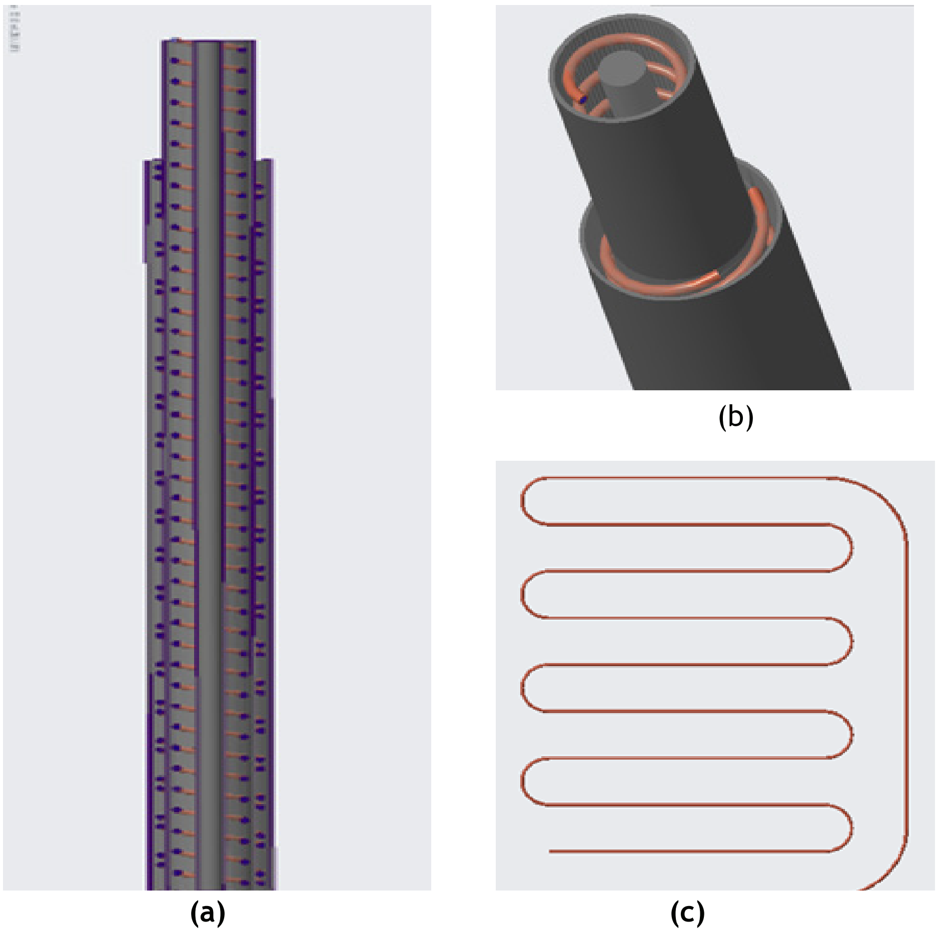

Figure 1. Models assembled made in Creo. (a) full cross-sectional view of Heat Exchanger, (b) heat exchanger with inlet and outlet. (c) condenser tubing used in the refrigerator.

model like Heat exchanger becomes heavy to operate, coz in meshing it generates too much nodes which means too many calculations and increment in Memory unit, file become heavy for computer to run and get slow to operate. Solution for this is to divide heat exchanger in multiple parts and run the analysis.

3.1 Design Modeler

"Design Modeler is a Workbench application from ANSYS that provides modeling functions unique for simulation that includes detailed geometry creation, CAD geometry modification simplification and concept model creation tools".

3.2 Meshing

"Meshing is often used in software-based simulation for Finite Element Analysis (FEA) and Computational Fluid Dynamics (CFD). It can significantly impact the accuracy of the simulation and the resources required to perform the simulation" (Table 1 and 2).

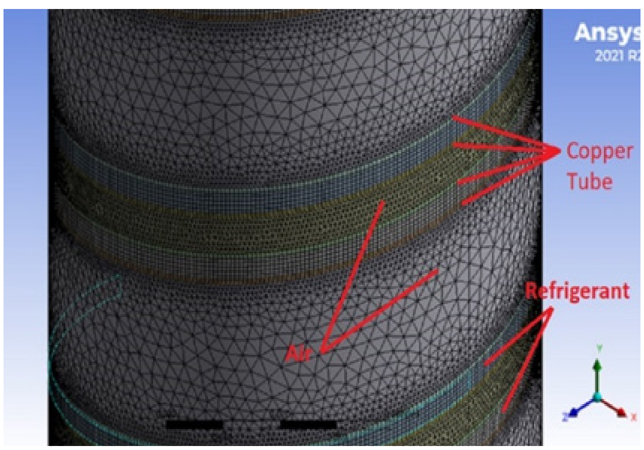

Figure 2. Magnified view of Fine Mesh indicating element location.

Table 1. Mesh Size from ANSYS report

Cells	Faces	Nodes
2193419	6653865	1261924

This is a meshing part of analysis report taken from the ANSYS software. The information related to the meshed cells, nodes is given along with the mesh quality. Minimum orthogonal quality and maximum aspect ratio is basic information in the mashing quality. Graphical representation about orthogonal meshing quality is shown in the Figure 2.

3.3 Realizable k-∈ Model

The realizable k-∈ model is a relatively recent development and differs from the standard k- ∈ model in two important ways:

- The realizable k-∈ model contains a new formulation for the turbulent viscosity.
- A new transport equation for the dissipation rate, ∈, has been derived from an exact equation for the transport of the mean-square vorticity fluctuation Figure 3.

In this case it can be clearly observed that Continuity and Energy is not converges during 110 iterations. It can also be checked in the graph plotted against these properties or parameters Figure 4. Plots of all properties should come or meet in one region, in that case it can be conclude that, the properties are convergent. Here in the controls if value of energy changes to 1e-05 from 1e-06, then energy will also be converged along with the other properties. Continuity equation will need much more iterations to get converged, nearly it will take 1060 iterations. Generally, in all problem continuity is very difficult to get converge with other properties.

Table 2. Mesh Quality from ANSYS report

Name	Type	Min Orthogonal Quality	Max Aspect Ratio
air_fluid	Tet Cell	0.011865018	23.797699
c1_fluid	Hex Cell	0.87688716	2.615556
c2_fluid	Hex Cell	0.85371216	2.7818073
c1_pipe	Hex Cell	0.63832216	3.8495139
c2_pipe	Hex Cell	0.65599242	4.1072188

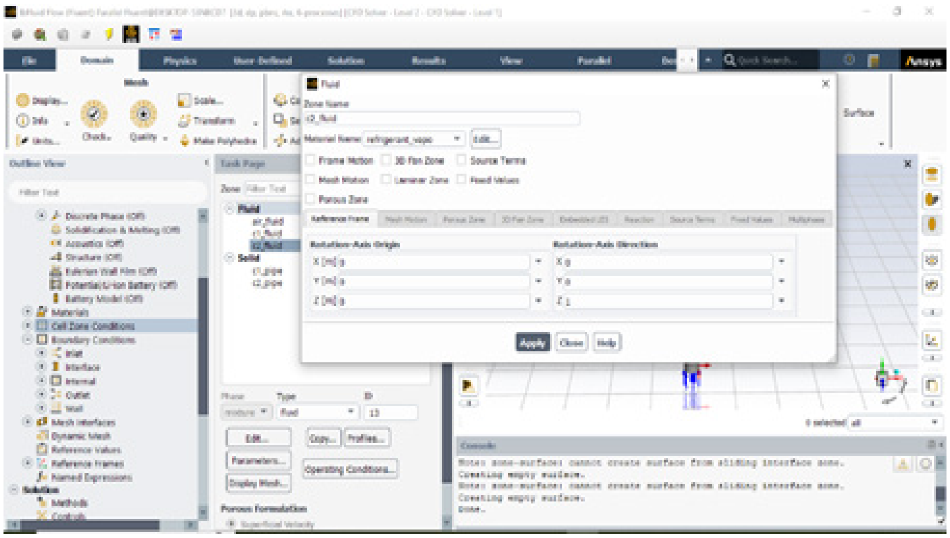

Figure 3. Boundary conditions setting.

3.4 Plots of Residuals

The density plot, velocity vector plot can be studied for the same case. As one of them is shown blow in Figure 5. In this Figure 5 flow of air is shown. In this case flow off air is in upward direction. Cold air is coming in through the bottom end, flowing around with the copper tubes and going out or leaving condenser section as a hot air. Here the plot is of velocity vector, showing vectors of velocity if air flow around copper tube and inside the condenser section. Here in the Figure 5 above two projects can be seen, project 'A' and project 'B'. Project A is one of the cases of experiment in which condenser section is studied only. In Project A variables are air velocity and the relative humidity are considered, where V_{air} is 9 m/s and RH is 60%. In next case only, the change is of variables. For second case variables V_{air} is 9 m/s and RH is 80% are considered to study. So, directly project 'B' is started and meshing data is transferred to it. In the Figure 6 it can be clearly seen that the data of meshing is directly provided the setup of the new project.

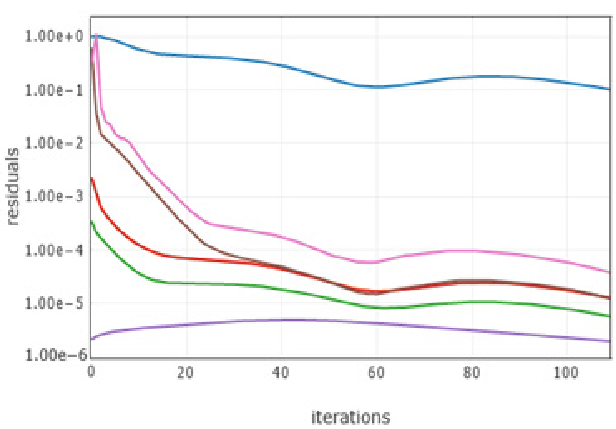

Figure 4. Graphical presentation of iteration vs. residuals.

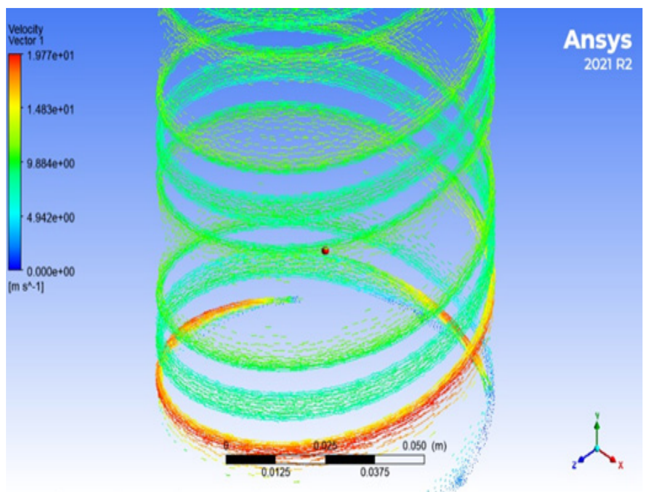

Figure 5. Velocity Vectors of flow of air.

4.0 Results and Discussion

Various cases have been studied in Ansys software using Fluent module. The outputs are observed in the post-processer. As per the output values received from the post-processor of Fluent are tabulated and the final

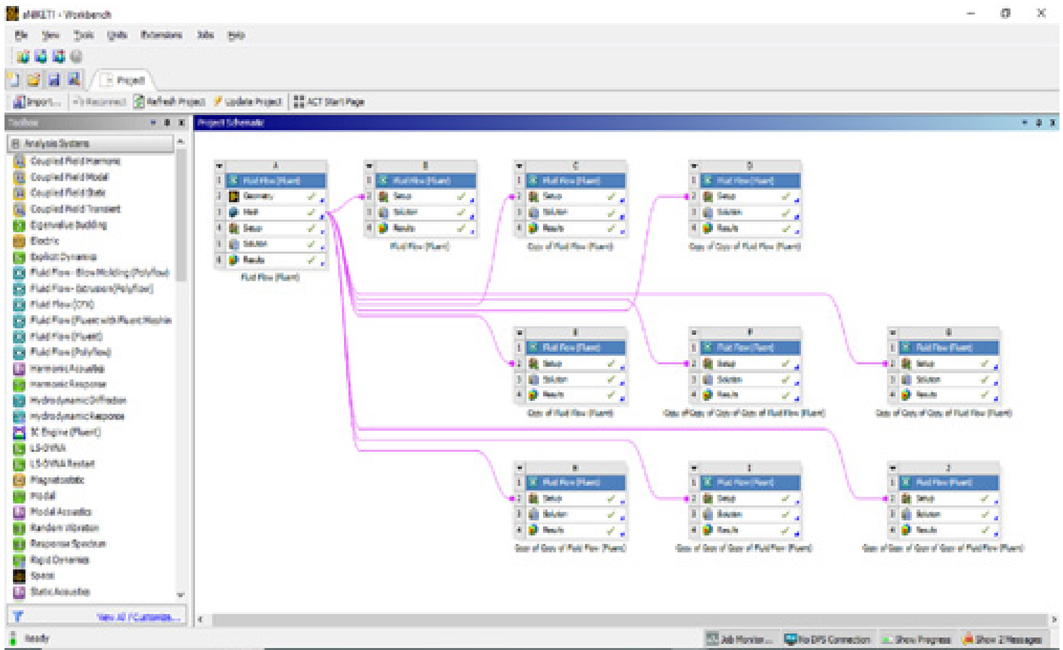

Figure 6. Multiple cases solved with same setup.

outputs are calculated. Calculated outputs are in the form of the COP of system, heat exchanger efficiency and mass of water collected.

4.1 COP of AWG

The study of AWG is done in CFD in 2 steps, first condenser section is studied and then evaporator section is studied. The input parameters are taken from the experimental values as a boundary condition. From the output of each case COP of AWG is calculated.

In the Figure 7, blue line indicates the COP of AWG in various 27 cases as mentioned before. Form the Figure 7 it van be observed that, 24th case has given highest COP 1.26. General, range of COP lie between 1.127 to 1.260.

4.2 COP of Refrigerator in Various Condition

As mentioned above the various cases have been studied in CFD for refrigerator. For the sake of comparison, input data for the for the boundary conditions is taken from the

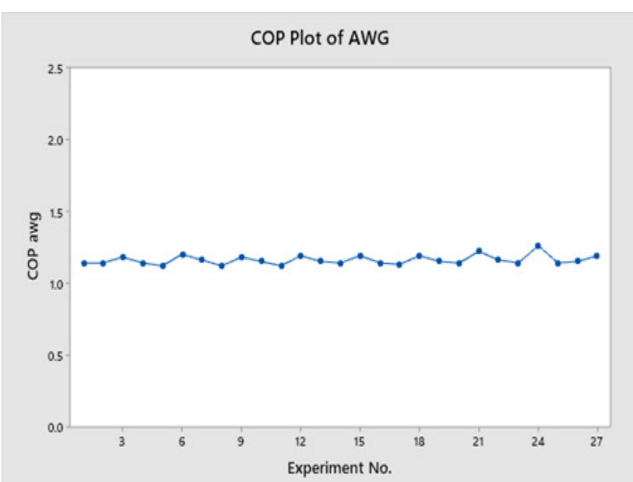

Figure 7. Graphical representation of Experiment no. Vs. COP of AWG

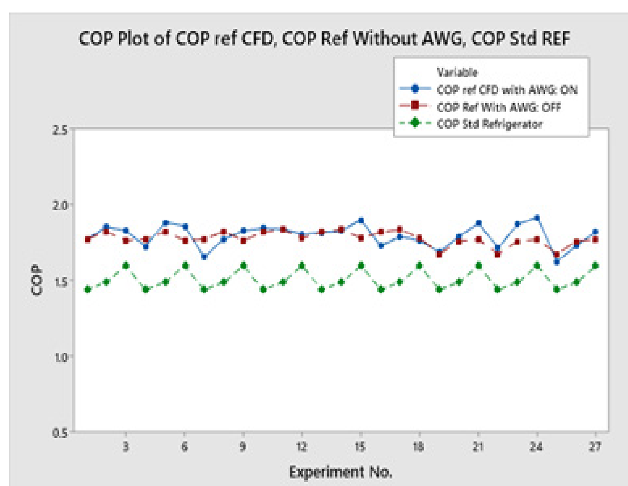

Figure 8. Comparative Graphical representation of Experiment no. Vs. COP of Refrigerator.

experimental values recorded. Here COP of refrigerator in various conditions are compared with each other. In the Figure 8 Green colour indicates the values of COP of standard refrigerator. There are only three cases of it for room temperature 20^0C, 25^0C and 30^0C only. But for comparison cases are repeated and shown 9 times with respect to parameter. Blue colour line indicates the COP of Refrigerator when its condenser is inside of heat exchanger of AWG. For this case both AWG and refrigerator are in working. Brown colour line indicates the COP of Refrigerator when its condenser is inside of heat exchanger of AWG but, AWG is not working. Only flow of air is there in heat exchanger.

In the graph it is clearly visible that COP of Standard Refrigerator remain between 1.437 to 1.593. Also, it means COP of other two cases are more. As far as COP of Refrigerator concern, COP with AWG is heist i.e. 1.91. But it cannot be clearly state that, COP with AWG is more than COP without AWG.

5.0 Conclusion

In this research study, a system is designed to separate the moisture from moist air through condensation process using VCR cycle. Ansys fluent is used here to test the designed system and to check the system performance.

Results of CFD study gives the clear understanding about the effective ness of result. System is able to extract the 95% to 100% moisture for air velocity of 7m/s and 9 m/s. For more velocity l1 m/s moisture condensation efficiency reduces up to 84%. For this level of velocity in heat exchanger average COP obtained of VCR system around 1.2 (as the standard value provided is 1.2) can be seen in graph.

Due to use of AWG with refrigerator, COP of refrigerator increases in the range of 1.437 to 1.593 can be seen in last graph mentioned. It clearly states that use of AWG and refrigerator together can improve COP if Refrigerator.

6.0 References

1. Das A, Das RS, Das K. Numerical analysis of liquid desiccant dehumidification system with novel trapezoidal baffled surface. Int J Refrig. 2023; 145:457-466. https://doi.org/10.1016/j.ijrefrig.2022.09.027
2. Li C, Ho H, Yang T, Amani M, Yan W. Numerical Simulation of a Flat-Sheet Membrane-based Dehumidifier with Various Serpentine Flow Channels. Int J Refrig. 2023; 145:50-58. https://doi.org/10.1016/j.ijrefrig.2022.09.003
3. Jadav C, Chowdhury K. Determining the appropriate fin spacing for high performance and low footprint of direct contact ambient air-based vaporization systems. Int J Refrig. 2023. https://doi.org/10.1016/j.ijrefrig.2023.02.015
4. Inamdar HV, Groll EA, Weibel JA, Garimella SV. Air-side Fouling of Finned Heat Exchangers: Part 1, Review and Proposed Test Protocol. Int J Refrig. 2023. https://doi.org/10.1016/j.ijrefrig.2023.02.017
5. Nguyen CK, Teodosiu C, Kuznik F, David D, Rusaouën G. Full-scale experimental study of moisture condensation on the glazing surface: condensation rate characterization. IOP Conf Ser Mater Sci Eng. 2019; 609(3):032035. https://doi.org/10.1088/1757-899x/609/3/032035
6. Roumeliotis I, Mathioudakis K. Analysis of moisture condensation during air expansion in turbines. Int J Refrig. 2006; 29(7):1092-1099. https://doi.org/10.1016/j.ijrefrig.2006.03.001
7. Raveesh G, Goyal R, Tyagi S. Parametric analysis of atmospheric water generation system and its viability in Indian cities. Thermal Sci Eng Prog. 2023; 39:101682. https://doi.org/10.1016/j.tsep.2023.101682
8. Ansari E, Ferber NL, Milošević T, et al. Atmospheric water generation in arid regions – A perspective on deployment challenges for the Middle East. J Water Process Eng. 2022; 49:103163. https://doi.org/10.1016/j.jwpe.2022.103163
9. Abd Elwadood SN, Dumée LF, Al Wahedi Y, Al Alili A, Karanikolos GN. Aluminophosphate-Based adsorbents for atmospheric water generation. J Water Process Eng. 2022; 49:103099. https://doi.org/10.1016/j.jwpe.2022.103099
10. Lovis L, Tremain P, Maddocks A, Moghtaderi B. Modelling of atmospheric water generation using desiccant coated heat exchangers: A parametric study. Energy Convers Manage. 2023; 279:116746. https://doi.org/10.1016/j.enconman.2023.116746
11. Shafeian N, Ranjbar A, Gorji TB. Progress in atmospheric water generation systems: A review. Renew Sustain Energy Rev. 2022; 161:112325. https://doi.org/10.1016/j.rser.2022.112325
12. Ni F, Xiao P, Zhang C, Chen T. Hygroscopic polymer gels toward atmospheric moisture exploitations for energy management and freshwater generation.

Matter. 2022; 5(9):2624–2658. https://doi.org/10.1016/j.matt.2022.06.010

13. Srivastava S, Yadav A. Water generation from atmospheric air by using composite desiccant material through fixed focus concentrating solar thermal power. Solar Energy. 2018; 169:302-315. https://doi.org/10.1016/j.solener.2018.03.089
14. Wang J, Liu J, Wang R, Wang L. Experimental investigation on two solar-driven sorption based devices to extract fresh water from atmosphere. Appl Therm Eng. 2017; 127:1608-1616. https://doi.org/10.1016/j.applthermaleng.2017.09.063
15. Patel J, Patel K, Mudgal A, Panchal H, Sadasivuni KK. Experimental investigations of atmospheric water extraction device under different climatic conditions. Sustain Energy Technol Assess. 2020; 38:100677. https://doi.org/10.1016/j.seta.2020.100677
16. Kandeal A, Joseph A, Elsharkawy M, Elkadeem M, Hamada MA, Khalil A, Eid Moustapha M, Sharshir SW. Research progress on recent technologies of water harvesting from atmospheric air: A detailed review. Sustain Energy Technol Assess. 2022; 52:102000. https://doi.org/10.1016/j.seta.2022.102000