Power Generation by Solar Chimney: Numerical and Experimental Investigation on Its Performance

Avvaru Sreekanth and P. Mallikarjuna

1 M.Tech. (MECH.), Dept of Mech., Annamacharya Institute of Technology and Science, Kadapa
2 Assistant Professor, Dept of Mech., Annamacharya Institute of Technology and Science, Kadapa

Abstract—Heat Exchangers play an important role in the development of refrigeration systems. However, the effectiveness of the heat exchangers depends on the geometry, working fluids, type and arrangement of the flow through heat exchangers. Various researchers around the world are working upon the enhancement of effectiveness through the identification of various working fluids. Further, the flow pattern through the heat exchangers has been given attention to enhance the wetting area of the fluid with solid components of the heat exchanger. In addition, micro-heat exchangers developed till date employ twisted tape inserts, baffles and fins to increase the surface area of the heat exchangers still keeping the effective dimensions of the heat exchangers to be low.

In the present work, a double pipe heat exchanger with corrugated pipes is proposed to enhance the thermohydraulic performance of the heat exchangers. The inner corrugated pipe carries the hot fluid and the annulus region carries the cold fluid. As the surface area of contact is increased, the heat transfer rate is expected to be increased. An investigation is done in the present work to find the pressure drop and heat transfer in the double pipe heat exchanger (DPHE) using Computational Fluid Dynamics (CFD). A commercial computational code namely ANSYS is used for the purpose of solving the general governing equations using finite volume method. Mass flow rates considered for the analysis are according to the practical conditions employed in industrial refrigeration systems. The operating temperatures of the DPHE considered in the present work are matching with the practical conditions. Various case studies were done in the present work to optimize the mass flow rate of the refrigerant through the DPHE. Further, due to the corrugations in the pipes, the rise in pressure drop expected. This pressure drop in the DPHE would contribute to the performance of the compressor in the refrigeration system thereby reducing the Coefficient of Performance (COP) of the system. Friction factors and Nusselt Numbers applicable to corrugated Double Pipe Heat Exchangers (DPHE) are investigated.

I. INTRODUCTION

In any process industry, we need to transfer heat for different operations (like cooling, heating, vaporizing, or condensing) to or from various fluid streams in various equipment like condensers, water heaters, reboilers, air heating or cooling devices etc., where heat exchanges between the two fluids. In a chemical process industry, the heat exchanger is frequently used for such applications. A heat exchanger is a device where two fluids streams come into thermal contact in order to transfer the heat from hot fluid to cold fluid stream. In this section, the technical analysis of the heat exchangers along with the method for predicting heat exchanger performance and operational parameters has been discussed. However, discussions on the economics (though discuss the heat exchanger size) of the heat exchanger have not been considered. In general heat exchangers may be categorized into two general classes depending on the relative orientation of the flow direction of the two fluid streams. If the two streams cross one another in space, usually at right angles, the heat exchangers are known as cross flow heat exchanger as shown in the Figure 1.

Figure 1 Schematic of cross flow heat exchanger
In the second class of heat exchanger the two streams move in parallel direction in space. The usual shell and tube heat exchanger or concentric pipe exchanger or double tube exchanger is the most frequently used exchanger in the class. Two situations may arise when the fluid flow in the parallel direction (Figure 2), one in which the fluids flow in same direction and the other in which the fluids flow in opposite direction (Figure 4). “Parallel flow” or “Co-current flow” is used when the fluid flow in same direction and counter current is used when the fluid flow is in the opposite direction. Before understanding the principle of heat exchanger we would first understand it from the point of construction.

![Figure 1 2 Schematic of parallel flow heat exchanger](image1)

**Figure 1 2 Schematic of parallel flow heat exchanger**

![Figure 1 3 Temperature profile along the length of the counter flow heat exchanger](image2)

**Figure 1 3 Temperature profile along the length of the counter flow heat exchanger**

![Figure 1 4 Schematic of perpendicular flow heat exchanger](image3)

**Figure 1 4 Schematic of perpendicular flow heat exchanger**

As the heat exchangers are tubular in nature (Note: we are not discussing about plate type heat exchangers). Thus we can easily find out the overall heat transfer coefficient based on our previous knowledge. Figure 6, shows a simplest form (double pipe heat exchanger) of tubular heat exchanger, where fluid A is being heated by fluid B in a co-current flow pattern. The inside and outside radii of the inner tube is represented \( r_i \) and \( r_o \). The length of the exchanger for heat transfer is considered as \( L \) for section 1 to 2. Thus the rate of heat transfer from the hot fluid to the cold fluid will be represented by

\[
q = \frac{T_A - T_B}{h_i \ln \left( \frac{r_o}{r_i} \right) + \frac{1}{h_A} + \frac{1}{h_B}}
\]

The overall heat transfer coefficient; Based on inside area of the inner pipe

\[
U_i = \frac{1}{h_i + \frac{A \ln \left( \frac{r_o}{r_i} \right)}{2\pi k L} + \frac{A}{h_A}}
\]
For the counter flow heat exchanger, the above two set equations show that the maximum possible heat exchanger is determined in terms of the inlet parameters. The maximum possible heat exchange may be determined (eq.8.6) by the fluid stream having low heat capacity rate:

\[ q_{\text{max}} = \frac{1}{U_o} \left( T_{hi} - T_{ci} \right) \]

Thus for the counter flow exchanger, the above two set equations show that the maximum possible heat exchanger is determined in terms of the inlet parameters. The maximum possible heat exchange may be determined (eq.8.6) by the fluid stream having low heat capacity rate:

\[ q_{\text{max}} = \frac{1}{U_o} \left( T_{hi} - T_{ci} \right) \]

The subscript ccf denotes counter current flow.

The ( ) is for the fluid having lower value of ( ).

Heat exchangers are usually cheaper when they have a smaller shell diameter and a long tube length. Thus, typically there is an aim to make the heat exchanger as long as physically possible whilst not exceeding production capabilities. However, there are many limitations for this, including space available at the installation site and the need to ensure tubes are available in lengths that are twice the required length (so they can be withdrawn and replaced). Also, long, thin tubes are difficult to take out and replace.
II. LITERATURE SURVEY

The literature survey has been done with regards to the existing methods or technologies available in order to increase the heat transfer rates in the heat exchangers. From the survey, it has been noticed that there are numerous studies available in which methods have been adopted to increase the heat transfer rates in circular double pipe heat exchangers. Very few studies are available in literature where corrugated geometry has been employed in order to investigate the heat transfer rates. Few studies involved nano-fluids those have been used to increase the rate of heat transfer. This section consist the various strategies that have been employed in the enhancement of heat transfer rates in the heat exchanger technology: Saeedan et al. [1] have investigated numerically the thermal performance of a helically baffled heat exchanger combined with a 3D fined tube operated with nano-fluids. At different volume concentrations nano-particles of Cu, CuO, and CNT have been considered in water based nano-fluids. It has been studied that how heat transfer and pressure drop are get affected by the Reynolds number and volume concentration. It has been found from the results that both heat transfer and pressure drop increases with the increase in the volume concentration and Reynolds number. The value of Nusselt number increases with the increase in volume concentration for CuO/water and Cu/water nano-fluid, while for CNT/water, it decreases as the volume concentration increased.

Sheikholeslami et al. [2] have done numerical and experimental investigation on turbulent flow and heat transfer in a double pipe air to water heat exchanger has been done using conical ring. Two arrays (Direct conical ring (DCR) array and Reverse conical ring (RCR) array) are considered. Experimental analysis has been done after considering different values of Reynolds number (6000-12,000), open area ratio (0-0.0833), conical angle (0°-30°) and pitch ratio (1.83-5.83). Statistical analysis in order to generate a correlation for friction factor, Nusselt number and thermal performance factor has been done. Also, to attain the optimal design, Non Sorting Genetic Algorithm II has been applied. Finite volume method has been exploited for numerical section in order to predict physical behaviour. It has been noticed from the results that the Nusselt number found to decrease with increase in open area ratio and pitch ratio when it expands with enhances of Reynolds number. Also, it has been found that friction factor reduces with boost of open area ratio, pitch ratio and Reynolds number. It has been concluded from the study that thermal performance rises with augment of conical angle for direct conical ring array.

Zarrella et al. [3] have done a relative analysis among a helical-shaped pipe and a double U-tube ground heat exchanger has been done. Thermal behaviour of the ground heat exchangers has been investigated when the two configurations were installed at a shallow depth and the interface among the ground surface and the ambient situation have been considered. By considering an equivalent electrical circuit of thermal resistances and capacitances, the difficulty related to heat transfer were solved. In order to comprise axial heat conduction in the ground and the borehole, same numerical simulation tool was used on a double U-tube heat exchanger. By considering the thermal properties of the ground, energy loads and the axial effects of weather conditions, two borehole heat exchangers were analyzed over both long- and short-term periods.

Demir et al. [4] did a computational investigation with constant wall temperature has been performed on forced convection flow of nano-fluids consisting of water with TiO2 and Al2O3 nano-particles in a horizontal tube. To determine the nano-fluid properties, Palm et al.’s correlations have been considered. In order to study the thermal and hydrodynamic behaviour of the nano-fluid flow, single-phase model having two-dimensional equations have been employed with either constant or temperature dependent properties. After the validation of the model by means of the experimental data of Duangthongsuk and Wongwises with TiO2 nanoparticles, the numerical analysis has been performed for a constant particle size of Al2O3 as a case study. The velocity and temperature profiles at the entrance and fully developed region have been drawn. The variations of the fluid temperature, local heat transfer coefficient and pressure drop along tube length have been presented. Effects of nanoparticles concentration and Reynolds number on the wall shear stress, Nusselt number, heat transfer coefficient and pressure drop have been presented. Numerical results show the heat transfer enhancement due to presence of the nano-particles in the fluid in
accordance with the results of the experimental study used for the validation process of the numerical model. Zhan et al. [5] introduced the thermodynamic analysis, based on the ideal gas state equation and energy conservation equation in order to appreciate the process principle of TVS simply. After validating the experimental results, the Kandlikar’s boiling heat transfer correlation is selected to predict the flow boiling process due to the low mass flow rate and low heat fluxes has been involved in flow boiling of the annular pipe fluid. In order to predict the heat transfer characteristics of double pipe heat exchanger under normal gravity, a quasi-steady state model has been recognized, with the bulk fluid natural convection, annular pipe two-phase boiling and inner pipe forced convection coupled from outside to inside. Established by the local pressure and temperature, it has been found that the fluid thermophysical properties are variable with the pipe length and time. Both the static analysis and the transient thermal performance of TVS heat exchanger are investigated with the variable fluid thermophysical properties.

Mehran et al. [6] have reported that instead of using cylindrical tube, conical tube has been employed as a novel improved geometry for double pipe heat exchangers. Various conical tube arrays with diverse flow directions were investigated. An inclusive study has been done in order to examine the effect of hydraulic, geometrical and thermodynamic individuality in heat exchanger. It has been found that entropy generation, entropy generation number, heat exchanger reversibility norm (HERN), heat transfer improvement number and effectiveness–NTU are the important concepts which have been considered for cases. The results show 55% and 40% increment in effectiveness and heat transfer improvement number at the optimum condition.

Han et al. [7] considered the process parameters, the characteristic numbers involving heat transfer characteristic, resistance characteristic and overall heat transfer performance calculated by CFD, and are served as objective functions to the RSM (Nusselt number for corrugated tube (Nuc), Nusselt number for smooth tube (Nus), fanning friction factor for corrugated tube (fc), Nuc/Nus, fc/fs and overall heat transfer coefficient (η)). The results of optimal designs are a set of multiple optimum solutions, called 'Pareto optimal solutions'. According to the Pareto optimal curves, the optimum designing parameters of double pipe heat exchanger with inner corrugated tube under the constrains of Nuc/Nus ≥ 1.2 are found to be P/D = 0.82, H/D = 0.22, r/D = 0.23, Re = 26,263, corresponding to the maximum value of η = 1.12.

Appadurai et al. [8] have studied the heat transfer using nano fluids in a double pipe heat exchanger which has been computed through Computational Fluid Dynamics (CFD) approach. Heat transfer performance of an internal fin in a circular tube has been experimentally scrutinized. For different Reynolds number ranging from 2.0x104 to 5.0x 104, wall temperature, bulk fluid temperature, and pressure drop along the axis of the finned tube were measured. It has been found from the study that there is increase of the thermal performance of nano fluids compared to water.

Huminic et al. [9] have carried out a 3-D analysis in to study the heat transfer characteristics of a double-tube helical heat exchangers using nanofluids. Nanoparticles with volume concentrations of 0.5–3 vol.% like CuO and TiO2 having diameters of 24 nm dispersed in water has been used as the working fluid. The mass flow rate of the water from the annulus was set at either half, full, or double the value and the mass flow rate of the nano-fluid from the inner tube was kept constant. Nano fluids and water temperatures variations along with heat transfer rates and heat transfer coefficients at the inner and outer tubes have been shown. For the same mass flow rate through the inner tube and annulus, it has been found that the heat transfer rate of nano fluid was roughly 14% greater than the pure water when 2% of CuO nano-particles were added in the water. From the study, it has been concluded that the convective heat transfer coefficients of the nano-fluids and water is found to increase with increasing of the mass flow rate and with the Dean number. The results have been validated by comparison of simulations with the data computed by empirical equations.

Sivakumar et al. [10] investigated the performance of heat transfer and effectiveness of the double pipe heat exchanger with two flow directions (one is parallel flow and counter flow). A commercial CFD package (ANSYS) has been used for this study to generate the 3D model heat exchanger. As a final point the
experimental assessment is validated with the numerical values.

III. SCOPE OF THE STUDY

The prime objective of the present study is to investigate the flow behavior such as pressure drop and friction factor in outer corrugated pipe along with the heat transfer between fluids. It is necessary to investigate heat transfer through both fluids for further calculations of heat exchanger thermal efficiency.

The specific objectives of the study are:

- To investigate pressure drop and friction factor in double pipe heat exchanger so that pumping power can be estimated.
- To visualize turbulence behavior in the outer corrugated pipe.
- To investigate velocity profiles at the boundaries (inlet and outlet).
- To examine maximum heat transfer rate through cold and hot fluid.

IV. RESULTS

Analytical Calculation

Equation 1 to Equation 5 represent the mathematical expressions that have been used in order to evaluate friction factor, Reynolds’s number, pressure drop, pumping power and Nusselt number. Using the post-processing module of the FLUENT, the shear stresses at the walls of the corrugated steel pipe are retrieved. These values are used in the calculation of friction factors at different flow rates for each type of the corrugated steel pipe using the Equation 1.

\[ f = \frac{8\tau_{wall}}{\rho V_{avg}^2} \]  

Reynolds Number can be estimated as

\[ \text{Re} = \frac{DV_{avg} \rho}{\mu} \]  

Where \( V_{avg} \) = Average velocity of flow in the pipe (m/s) and \( \rho \) = Density (kg/m3). Friction factors values calculated from Equation 3 are used to estimate the pressure drop at different flow rates for each of the case using

\[ \Delta p = \frac{f L V^2}{2D_h \rho} \]  

(3)

Where \( \Delta p \) = Pressure Drop (Pa), \( f \) = Friction Factor, \( L \) = Length of the HTS cable (m), \( D_h \) = Hydraulic Diameter (m).

The pumping power needed to pump the water through the corrugated steel pipe of HTS cable is calculated using the pressure drop values estimated from Equation 4.

\[ W = \Delta p V \]  

(4)

\[ V = A \times V_{avg} \]

(5)

Where \( V_{avg} \) is volume flow rate (L/min) and A is the area of cross-section of the pipe (m2).

Now, Nusselt number, a dimensionless form of the temperature gradient in the fluid at the heat transfer surface, is calculated from Equation 5.

\[ Nu = \frac{h L}{k} \]  

(6)

Where, \( h \) = Heat Transfer coefficient (W/m2-K), \( k \) = thermal conductivity (W/m-K). The unknown value, the heat transfer coefficient is estimated from Equation 6 at different flow rates for each corrugation pitch of fixed depth
the fluid. The different values of wall temperatures are obtained from the numerical solution.

In the present study, pressure drop analysis has been done in order to evaluate the turbulent flow effects when hot water is flowing in the inner corrugated pipe of double pipe heat exchanger. Figure 4.1 shows the contour plot for total pressure in DPHE. Figure 4.2 shows the contour map of total pressure at outer corrugated wall of the DPHE.

Figure 4.1: Contours plot of total pressure in a plain DPHE

Figure 4.2: The contour map of total pressure on outer corrugated wall

Figure 4.3 shows the pressure drop along the axis of the double pipe heat exchanger. It can be noticed that for a length of 1000mm there is a 4-5Pa pressure drop. Figure 4.4 and Figure 5.5 shows pumping power required to pump the hot and cold fluid. It can be noticed that the pumping power found to increase with the increase in mass flow rate for both hot and cold fluid.

Figure 4.4 Pumping power vs. Mass flow rate for hot fluid

Figure 4.5 Pumping power vs. Mass flow rate for cold fluid

Figure 4.6 Friction factor vs. Reynolds Number variation for inner corrugated pipe
Figure 4.7 Friction factor vs. Reynolds Number variation for outer corrugated pipe

Figure 4.8 and Figure 4.9 show the velocity variation along the cross-section of the hot fluid domain at inlet and outlet respectively. It can be noticed that the velocity at the outlet (Figure 4.9) of the corrugated pipe is still in the development phase. It means that the length is small due to which the fluid has not enough distance to travel within the DPHE so that it can attain a fully developed flow at the outlet of the inner corrugated pipe. It can be observed that as the mass flow rate increases the velocity of the fluid is found to increase in order to maintain the continuity of the flow. Also, it can be examined that at the outlet of the inner pipe the fluid velocity is more as compared to at inlet section of the pipe where hot fluid is meant to enter.

Figure 4.8: Velocity profiles at inlet for inner corrugated pipe

Figure 4.9: The contour map of total temperature inside the heat exchanger

Figure 4.10 contour map of turbulent kinetic energy at inner wall of DPHE

Figure 4.11 shows the variation in Nusselt number with the Reynolds number for cold and hot water flowing through the outer and inner corrugated pipes of double pipe heat exchangers. It can be observed from the plots that here the Nusselt number is found to increase with the increase in the Reynolds number. This may be due to the increase in the convective heat transfer coefficient which leads to increase in total heat transfer rates.

Figure 4.11: Variation in Nusselt number with Reynolds number
V. CONCLUSION

In the present study, computational analysis on the corrugated double pipe heat exchanger has been done using ANSYS FLUENT 15.0. Corrugated fluid domains were created in the software and turbulent flow analysis has been done using Realizable 2 Equation Viscous Model after generating coarse meshing. The analysis was performed by considering different mass flow rates of cold and hot fluid as given. As Nusselt number is found to increase with Reynolds number therefore it can be concluded that the total heat transfer rates will increase with the increase in the convective heat transfer coefficients.

REFERENCES


