A Comparative Computational Analysis on Performance of a Heat Exchanger with Coil Inserts of Various Pitch Lengths

Firoz Alam[1], Yasir Baig[2], Sanjay Singh[3]

[1]*M.E. Scholar, Department of Mechanical Engineering, SISTec, Bhopal, India*
[2] *Assistant Professor, Department of Mechanical Engineering, SISTec, Bhopal, India*
[3]*Associate Professor, Department of Mechanical Engineering, SISTec, Bhopal, India*

---***---

Abstract - *In present research work the problem of variation of heat transfer rate along with other performance parameters in simple tube in tube heat exchanger (generally used in practice) is compared with tube in tube heat exchanger having coil inserts of various pitch length inside a tube in tube heat exchanger with the help of a computer simulation performed on an analysis software, namely ANSYS-Fluent 16.0. The inlet conditions of all the cases are kept constant for all three systems and variations in outlet parameters are noticed with the help of simulation results as provided by the software. An increase in performance is noted with the introduction of inserts in the heat exchanger and the variation of results is also seen with varying pitch length of inserts. It is found that for the same geometric and kinematic parameters the heat transfer rate of the helically twisted tube with circular cross section and 6.5 inch pitch length, and is maximum at a volume flow rate of 105LPH with a value of 2880.8 Watts which is 4.9% greater than the heat transfer rate achieved in the twisted insert with 2.5 inch cross section and it is 10.59 % greater than the heat transfer rate achieved in plane tube at 105LPH. it is found out that the overall heat transfer coefficient is having a maximum value of 3117.2 Watts/m2k for the twisted circular cross sectional insert of 6.5 inch pitch length, which is 6.5% more than the value we obtained for insert with 2.5 inch pitch length and circular in cross section and 29% more than the value achieved for plane tube. As the results depict a significant improvement in the performance of a simple heat exchanger so this may prove to be very efficient in chemical industries as well as other sectors in which a fluid to fluid heat transfer takes place.*

***Key Words*:** Heat Exchanger; Heat Transfer Rate; Effectiveness; CFD; concentric tube…

1. INTRODUCTION

A heat exchanger is a device that transfers heat from a high thermal potential fluid to a low thermal potential fluid by either letting the fluids come in direct contact with each other or by separating them with a conductive material wall. Such devices have their utility in thermal power stations, paper industries, chemical industries and many more. These days energy crisis plays a key role on human life and utilizing the energy resources efficiently by improving heat transfer rate of heat exchanger has become a topic of high interest. A huge amount of research is going on the topic of enhancement of heat transfer rate techniques

The enhancement of heat transfer rate is noticed to be achieved by applying various active and passive methods by various researchers and this paper is about discussing and acknowledging their efforts in this field. The heat transfer rate can be increased by increasing either the surface area of heat exchanger or the overall heat transfer coefficient and in order to make the heat exchanger more compact, we are restricted with a given surface area and weight. The only option remains is to increase the heat transfer coefficient of the system

2. Literature Survey

The heat transfer process focuses on heat transfer rate, and on which a great amount of research work have been done in the form of research papers, book chapters and patents. A brief review of same has been presented in this section.

Shu-RongYan et al (2020) [1]: a In this study attempts are made to clarify some hidden features of effectiveness concept (the ratio of actual heat transfer to maximum possible heat transfer) of heat exchangers and provide a critique viewpoint and comprehensive description about that. It is shown in present paper that the effectiveness parameter requires much more attention when it is used in fluid-flow parametric or sensitivity studies of heat exchangers and lack of consideration of its hidden features may lead to incorrect or imperfect decision- makings on curve behavior of effectiveness. Indeed, it is shown that the curve trend of effectiveness against the Reynolds number of one side of heat exchanger can be ascending, descending, or ascending-descending depending on the mass flow rate (*Re* number) of the other side fluid.

J.I.Córcoles et al (2020) [2]: In this study, 3-D numerical simulations were carried out to analyze the influence of geometrical parameters for eight spirally inner corrugated tubes at turbulent flow (Re=25×103) in a double pipe heat exchanger. As a novelty, different combinations of pitch and height in a 3-D inward corrugated tube numerical model were analysed and validated with an experimental setup. This had not previously been conducted in a double pipe heat exchanger. Furthermore, the numerical model included the entire geometry of the heat exchanger, with dimensions of the computational domain similar to those used in actual commercial applications. Grid independence analysis of the numerical solution was performed based on a 3-D unstructured tetrahedral mesh scheme, considering the

Realizable k–ε turbulence model. Case 8, with the highest corrugation height (H/D=0.05) and the lowest helical pitch (P/D=0.682) presented the highest pressure drops in both inner and annular tubes, being 4.15 and 1.27 times higher in the inner tube and in the annulus side than in the smooth tube, respectively. Regarding heat transfer, Case 9, with the smallest helical pitch and an intermediate corrugation height (H/D=0.041) obtained the highest number of transfer units (NTU) value, which, under the experimental conditions of this work, Resulted in an increase of 29% compared with the smooth tube. In Cases 7 and 9, the inner tubes showed optimal results when considering the combined influence of the enhanced heat transfer performance and pressure drop using the performance evaluation criteria (PEC).

Miftah Altwieb et al (2020) [3]: New simulation and experimental results have been obtained and are presented for a multi-tube fin heat exchanger unit, from which semi-analytical correlations for the Fanning friction and Colburn factors were developed. The multi-tube and fin heat exchanger represents the main component of the Fan Coil Unit, an essential component of HVAC systems used for domestic and commercial heating and cooling. Improving the efficiency of the heat exchanger typically comes at the expense of higher pressure drops or costlier materials and production costs. Here, an experimental setup was designed and constructed to evaluate the thermal performance of such a heat exchanger. Geometrical modifications were explored for thermal performance enhancement. Furthermore, full three-dimensional CFD case studies of the heat exchanger were investigated to examine the effect of the geometrical features on the air side of the heat exchanger to study the effect of fin spacing, transverse and longitudinal pitches. The CFD model developed was first globally validated against experimental results. The model results were used to predict the Fanning and Colburn factors and the local fin efficiency based on the carefully selected geometric parameters. The data obtained was utilised to develop two new semi-analytical models for the Fanning and Colburn friction factors which were well within ±10% error bands and showed strong correlation coefficients of more than 98 and 97% respectively.

Zargoushi et al (2019) [4]: a CFD model in ANSYS FLUENT is developed for better understanding the transport phenomena, especially the phase change, in a quite complex plate-fin heat exchanger operated in a gas refining company. The flow channels, ducts and passes of the cold box, are considered in the computational geometry. The porous media technique is introduced in the computational domain due to the excessive increase in the number of computational grids when the fins accounted for in the numerical domain. Numerical results are compared with the corresponding operational data and the best agreement is obtained by the LTNE approach. The calculated mean relative errors of the LTNE approach for the outlet vapor fraction of streams A, hot stream, B and C, sided cold streams, are 0.15, 0.352, and 0.173%, respectively.

Chamil Abeykoon (2019) [5]: This study aims to investigate the design procedure of a heat exchanger theoretically and then its performance will be analyzed and optimized using computational fluid dynamics. For the design purposes, a counter flow heat exchanger was considered and its length was theoretically calculated with the LMTD method while the pressure drop and energy consumption were also calculated with the Kern method. Afterwards, a computational model of the same heat exchanger was implemented with ANSYS and then this model was extended to six different models by altering its key design parameters for the optimization purposes. Eventually, these models were used to analyze the heat transfer behavior, mass flow rates, pressures drops, flow velocities and vortices of shell and tube flows inside the heat exchanger. Theoretical and CFD results showed only a 1.05% difference in terms of the cooling performance of the hot fluid. The axial pressure drops showed positive correlations with both the overall heat transfer coefficient and pumping power demand. Overall, the results of this study confirms that CFD modeling can be promising for design and optimization of heat exchangers and it allows testing of numerous design options without fabricating physical prototypes.

P.C. Mukesh Kumar et al (2019) [6]: In this investigation, the heat transfer and pressure drop of the double helically coiled heat exchanger handling MWCNT/water nanofluids have been analyzed by the computational software ANSYS 14.5 version. The computational analysis was carried out under the laminar flow condition in the Dean number range of 1300– 2200. The design of new shell and double helically coiled tube heat exchanger was done by using standard designing procedure and 3D modeling was done in Cre-O 2.0 parametric. The Finite Element Analysis software ANSYS Workbench 14.5 was used to perform CFD analysis under the standard working condition. The MWCNT/water nanofluids at 0.2%, 0.4%, and 0.6% volume concentrations have been taken for this investigation. The major factors like volume concentrations of nanofluids and Dean Number are considered for predicting the heat transfer rate and pressure drop. The simulation data was compared with the experimental data. It is studied that the heat transfer rate and pressure drop increase with increasing volume concentrations of MWCNT/water nanofluids. It is found that the Nusselt number of 0.6% MWCNT/water nanofluids is 30% higher than water at the Dean number value of 1400 and Pressure drop is 11% higher than water at the Dean number value of 2200. It is found that the simulation data hold good agreement with the experimental data. The common deviation between the Nusselt number and pressure drop of CFD data and the Nusselt number and pressure drop of experimental data are found to be 7.2% and 8.5% respectively.

A. Natarajan et al (2019) [7]: Heat transfer enhancement using nano-fluids has gained significant attention over the past few years. Nano-fluids are potentially applicable as alternative coolants for many areas such as electronics, automotive, air conditioning, power generation and nuclear applications. Heat transfer coefficient and the friction factor characteristics of SiC/water nanofluid will have been numerically investigated using ANSYS FLUENT 14.0. The Nanofluid was employed in a circular tube equipped with modified Horizontal Wing Twisted Tapes (HWTT) with different twist ratio (y = 2.0, 4.4, 6.0) were used for simulation and compared with Plain Twisted Tapes (PTT). The results of CFD investigations of heat transfer Coefficient and friction characteristics are presented for the HW-TT with Different twist ratio in comparison with the P-TT case.

Robert Ștefan Vizitiu et al (2019) [8]: Recovery from residual hot water in the building sector. The purpose of this research is to improve the efficiency of a heat pipe heat exchanger by using a phase change material. The device can decrease the energy demands of a building and it can be a reliable alternative source to produce thermal energy. The device can decrease the energy demands of a building and it can be a reliable alternative source to produce thermal energy.

Jian Ge et al (2018) [9]: Passive residual heat removal heat exchanger (PRHR HX) is a key equipment in advanced passive safety pressurized water reactors (PWRs), such as AP1000. Immerged in the In-containment refueling water storage tank (IRWST), the C-shape PRHR HX removes the residual heat using natural convection and boiling heat transfer during postulated accidents. Therefore, the safety operation of the PRHR HX is very important for the nuclear power plant (NPP). In this paper, three dimensional CFD simulation of the secondary side fluid flow and heat transfer of the PRHR HX are performed. The drift flux model is used to simulate the two phase flow phenomenon in the IRWST. The tube region is modeled by the porous media approach. The flow resistance in the tube region is calculated by empirical pressure drop correlation and two phase flow multiplier. The heat transfer rate from the primary side to the secondary side fluid is also evaluated by widely used heat transfer empirical correlations. Additional source terms corresponding to the flow resistance and removed heat in the tube region are added to the momentum and energy equation, respectively. The governing equations are solved by the commercial CFD package FLUENT.

Jian Ge et al (2018) [10]: Passive residual heat removal (PRHR) system is a very important component of the passive safety systems in advanced passive safety pressurizer water reactors (PWRs) such as AP1000. The passive residual heat removal heat exchanger (PRHR HX) is a C-shape tube bundle heat exchanger immerged in the In-containment refueling water storage tank (IRWST) which removes the core decay heat during the accident transients. The performance of the PRHR HX is significant for the safety of the nuclear power plant (NPP). In this paper, the thermal hydraulics characteristics of the PRHR HX in the IRWST are analyzed using computational fluid dynamics (CFD). The tube region is modeled by the porous media approach along with the distributed resistance method. Heat transfer from the primary side fluid inside the tube to the secondary side fluid in the IRWST is considered. The simulation is carried out by the commercial CFD package FLUENT. The calculation of the flow resistance and heat transfer in the tube region is implemented using the User Defined Functions (UDF) in FLUENT based on the local flow conditions. Three dimensional distributions of the fluid velocity and temperature in the IRWST are obtained and thermal stratification is observed. The PRHR HX heat transfer capacity and the primary side fluid temperature distribution inside tubes are analyzed.

3. INTRODUCTION TO ANSYS

Similar to solving any problem analytically, you need to define:

(1) your solution domain,

(2) the physical model,

(3) boundary conditions and

(4) Physical properties.

Below describes the processes in word slightly additional adjust to the software system.

Virtual Modelling

Construct a two or three dimensional virtual model of your project for the representation of the object and test it using the work plane coordinates system within Fluent-ANSYS.

Assigning material

Now that the part is modelled, outline a library of the mandatory materials that compose the item (or project) being modelled. This includes thermal and mechanical properties

Generate Mesh

At this point ANSYS understands the makeup of the part. Now define how the Modelled system should be broken down into finite pieces in order to perform the calculation of a infinitesimal object and then the system will use the iterative method to integrate the results for the complete system.

Problem set-up

Once the system is fully designed, the last task is to set-up the system with constraints, such as physical loadings or boundary conditions. Here we will be providing flow rates of fluid, gravity effect and type of differential equation that the problem is depending upon.

Generate Solution

Here the software requires information about the type of analysis that it has to undertake (steady state/transient). The software also runs a sample of 10 iterations in order to check that whether the solution will converge to a unique value or not.

View Results/Reports (post Processing)

Post Processing is the technique of getting the solution represented in the format you desire. Ansys provides various ways in which the solution can be presented in, and you can choose from among them such as tables, graphs, and contour plots.

4. METHODOLOGY

The Computational methodology and procedure that is undertaken in order to perform the required analysis over the concentric tube heat exchanger. The method for calculating effectiveness, LMTD and overall heat transfer coefficient and their formula are discussed in here as well. Mathematical calculation and value gained for straight Copper tube concentric heat exchanger with coil inserts of various pitch lengths and their effects are observed.

COMPUTATIONAL PROCEDURE:

1) A computational fluid dynamic analysis is performed on ANSYS-Fluent (2016) software with different volume flow rates of hot fluid in counter flow arrangement to cold fluid. The volume flow rate of hot water is varied with values of 65,75,85,95 and 105 LPH respectively, whereas the volume flow rate of cold fluid is maintained at 65LPH.

2) The same temperature of hot and cold fluid at inlet was provided for all the simulations and results are checked in the form of Hot water and cold water outlet temperatures. The simulation was performed for the three different pitch lengths of coils that are inserted in a plane tube heat exchanger. The virtual models of which are also designed on the same software and the boundary conditions of all the simulations are kept same for performing the comparative analysis.

3) The system is then given the appropriate boundary conditions and made to perform its first set of calculations. The result is obtained after 1273 iterations and we were able to see the first set of readings in the form of outlet temperatures.

4) After taking the first set of readings, the flow rate of hot fluid was changed to the next value and the system is allowed to run calculations for the same boundary conditions until it reaches the steady state. The number of iterations that took for the system to reach to a steady state is around 1200 to 1500. Similar procedure was used for taking the readings with flow rates of 95,85, 75, 65LPH of hot fluid.

For every set of readings the system is allowed to run calculations for time duration of 4-5 hours in order to complete its iterations to reach to a steady state and provide a converged solution.

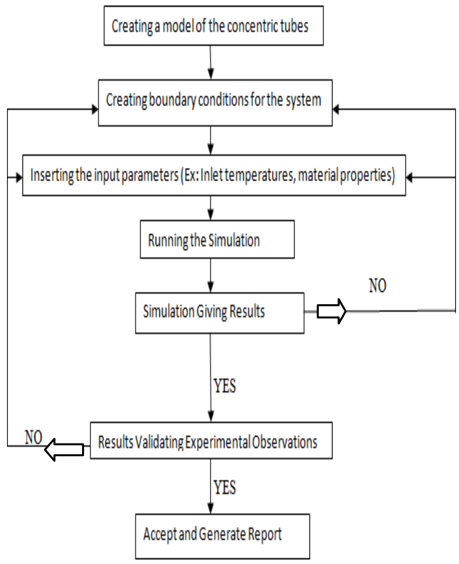
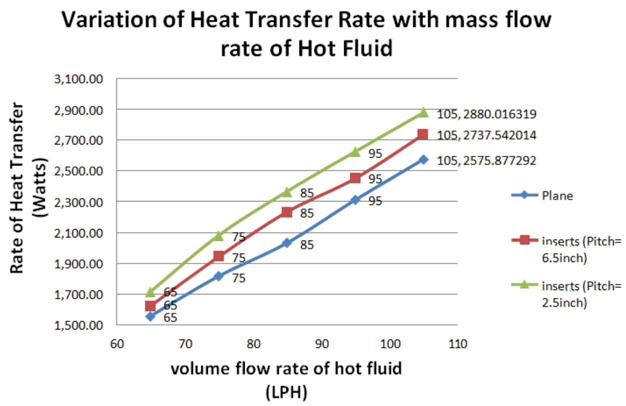


Fig 4. a

5. RESULTS AND DISCUSSIONS

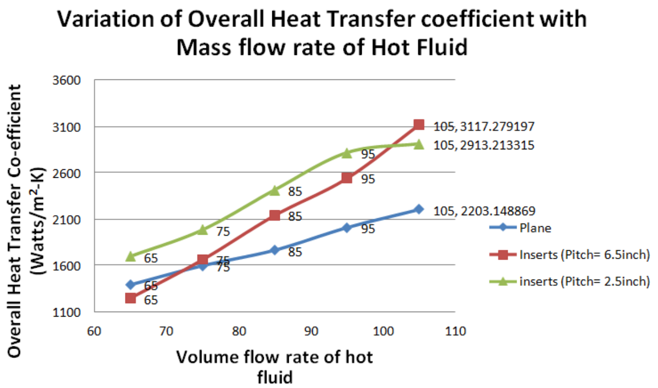
5.1 Comparison of Heat Transfer Rate of all the three cases

Table 5.5 and fig 5.1 shows the comparative values of heat transfer rates of concentric circular plane tubes without any inserts, tubes with helically twisted inserts of pitch 2.5 inch and tubes with helically twisted inserts of pitch 6.5 inch respectively. The cross-section of inserts is circular in shape. The comparison shows that the maximum value of heat transfer rate for the same flow rates is achieved for inserts with circular cross section and it was at its maximum on 105LPH with a value of 2880.0163 Watts.



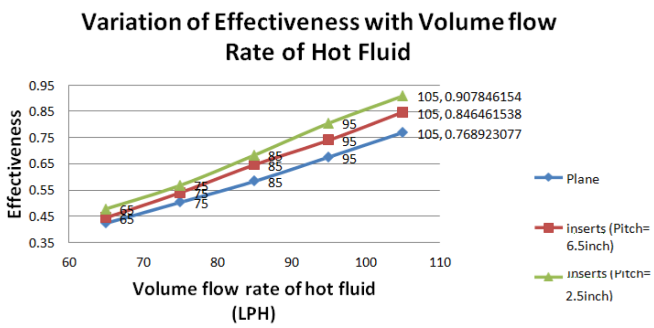
5.2 Comparison of Overall Heat Transfer coefficient based on Experimental analysis of all the three cases.

Fig 5.2 shows the comparative values of Overall Heat transfer coefficient of concentric circular plane tubes without any inserts, tubes with helically twisted inserts of pitch 2.5 inch and tubes with helically twisted inserts of pitch 6.5 inch respectively. The cross-section of inserts is circular in shape. The comparison shows that the maximum value of Overall Heat transfer coefficient for the same flow rates is achieved for inserts with circular cross section and 6.5inch pitch length. It was at its maximum on 105LPH with a value of 3117.279 (Watts/ (m^2-K)).



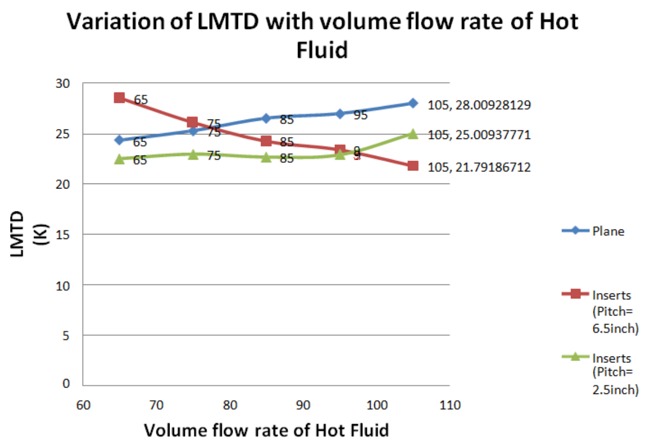
5.3 Comparison of Effectiveness based on Experimental analysis of all the three cases

Fig 5.3 shows the comparative values of Effectiveness of concentric circular plane tubes without any inserts, tubes with helically twisted inserts of pitch 2.5 inch and tubes with helically twisted inserts of pitch 6.5 inch respectively. The cross-section of inserts is circular in shape. The comparison shows that the maximum value of Effectiveness for the same flow rates is achieved for inserts with circular cross section and pitch length of 2.5 inches, and it was at its maximum on 105LPH with a value of 0.90784.



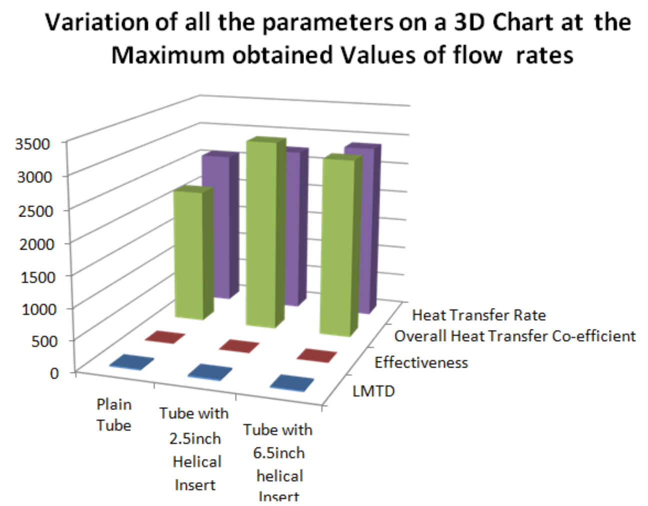
5.4 Comparison of LMTD based on Experimental analysis of all the three cases

Fig 5.4 shows the comparative values of LMTD of concentric circular plane tubes without any inserts tubes with helically twisted inserts of pitch 2.5 inch and tubes with helically twisted inserts of pitch 6.5 inch respectively. The cross-section of inserts is circular in shape. The comparison shows that the maximum value of LMTD for the same flow rates is achieved for inserts with circular cross section and it was at its maximum on 65LPH with a value of 28.524K.



5.5 Comparison of all the parameters based on Numerical analysis of all the three cases

Fig 5.5 shows the comparative values of all the discussed parameters on a single 3D chart. The values are taken at the maximum conditions of the cases discussed about.



6. CONCLUSIONS

1) In this research work it is found that for the same geometric and kinematic parameters the heat transfer rate of the helically twisted tube with circular cross section and 6.5 inch pitch length, and is maximum at a volume flow rate of 105LPH with a value of 2880.8 Watts which is 4.9% greater than the heat transfer rate achieved in the twisted insert with 2.5 inch cross section and it is 10.59 % greater than the heat transfer rate achieved in plane tube at 105LPH.

2) In the present research work it is found out that the overall heat transfer coefficient is having a maximum value of 3117.2 Watts/m2k for the twisted circular cross sectional insert of 6.5 inch pitch length, which is 6.5% more than the value we obtained for insert with 2.5 inch pitch length and circular in cross section and 29% more than the value achieved for plane tube.

3) It is noted that LMTD for circular cross sectional insert and 6.5 inch pitch length was found to be 28.52 K which is greater than 2.5inch pitch length insert by 21% and greater then plane tube by 14.43%.

4) The effectiveness of 2.5inch pitch length circular cross sectional insert was also found to be maximum with a value of 0.9078 which is more than that of 6.5inch pitch length circular cross section insert by 6.7% and plane tube by 15.3% respectively at the same flow rate of 65LPH of hot fluid.

5) The NTU which is a measure of physical size of the heat exchanger is maximum for 6.5 inch pitch length circular cross sectional twisted insert. So if we would require the same heat transfer rate from all the system then the required area for this case will be minimum.

REFERENCES

1.] Shu-Rong Yana, Hazim Moria , Samira Pourhedayat , Mehran Hashemian (2020). A critique of effectiveness concept for heat exchangers; theoretical-experimental study, International Journal of Heat and Mass Transfer (2020).

2.] J.I. Corcoles , J.D. Moya-Rico, A.E. Molina, J.A. Almendros-Ib anez (2020). Numerical and experimental study of the heat transfer process in a double pipe heat exchanger with inner corrugated tubes. International Journal of Thermal Science (2020).

3.] Miftah Altwieb, Krzysztof J. Kubiak, Aliyu M. Aliyu, Rakesh Mishra (2020). A new three-dimensional CFD model for efficiency optimisation of fluid-to-air multi-fin heat exchanger. Thermal Science and Engineering Progress (2020).

4.] A. Zargoushi, F. Talebi, S.H. Hosseini (2019). CFD modeling of industrial cold box with plate-fin heat exchanger: Focusing on phase change phenomenon. International Journal of Heat and Mass Transfer (2019).

5.] Chamil Abeykoon (2019). Compact heat exchangers – Design and optimization with CFD. International Journal of Heat and Mass Transfer 146 (2020) 118766.

6.] P.C. Mukesh Kumar, M. Chandrasekar (2019). CFD analysis on heat and flow characteristics of double helically coiled tube heat exchanger handling MWCNT/water nanofluids. Heliyon 5 (2019) e02030.

7.] A. Natarajan, R. Venkatesh, S. Gobinath, L. Devakumar, K. Gopalakrishnan (2019) CFD simulation of heat transfer enhancement in circular tube with twisted tape insert by using nanofluids. Materials Today: Proceedings (2019).

8.] Robert Ştefan Vizitiua, Andrei Burlacua, Dorina Nicolina Isopescua, Marina Verdeşa, Gavril Sosoia, Constantin Doru Lăzărescua. CFD analysis of an innovative heat recovery system. Proceedia Manufacturing 32 (2019) 488-495.

9.] Jian Ge, Wenxi Tian, Suizheng Qiu, G.H. SuCFD. simulation of secondary side fluid flow and heat transfer of the passive residual heat removal heat exchanger. Nuclear Engineering and Design 337 (2018) 27–37.

10.] Jian Ge, Wenxi Tian, Suizheng Qiu, G.H. Su. CFD investigation on thermal hydraulics of the passive residual heat removal heat exchanger (PRHR HX). Nuclear Engineering and Design 327 (2018) 139–149.