Numerical investigation of flow characteristics of a counter flow heat exchanger

Om Parkash

Department of Mechanical Engineering

Amity University Haryana-122413, India

Abstract- A numerical investigation of counter flow heat exchanger in a particular tube in tube heat exchanger with reversing the flow is investigated in this paper. The investigation includes the heat transfer and the pressure drop in the heat exchanger considering water as a fluid. Many researchers have reported regarding the enhancement of counter flow heat exchangers altering the fluid temperature and using different fluids. Here to foresee the outcome of the heat exchanger in terms of temperature change and pressure drop due to variation of temperature and mass flow rate a CFD program fluent has been used. A set of CFD simulation is carried out for the double pipe counter flow heat exchanger and after validating the methodology of CFD analysis the effects are established. The experimental results available in the literature [10] are compared with CFD fluent simulation results in ANSYS 14. We have got a better comparison with the previous results with net average temperature of 323.80K using aluminium as a material of construction and viscous k-ε model for the flow domain. However there is a very less marginal error of less than 2% comes in the simulation. The values and the results are case responsive to the turbulence model section.

Keywords- Counter flow heat exchanger, temperature change, RNG k-ε model

I INTRODUCTION

It has been widely admired that the counter flow heat exchanger has a greater heat transfer capability as compare to other heat exchangers. In counter flow heat exchanger the fluid flows inside the tube and kept apart from each other with a boundary wall where the heat transfers from one fluid to other without mingling. It has great advantage of very compact in size. Heat exchanger is designed and modelled using the correlation based analytical approach which means constantly used to improve the data periodically. Correlation approach is also applied for the sizing and used for getting better results after successive iteration. Although it is very difficult to get the right combination for the analysis, the CFD makes it easier for the calculation. The transfer of sensible heat takes place due to the significances of second law of thermodynamics. Different computational analysis has been adopted to analyse and configure the heat exchanger with some altercation like changing the inlet temperatures and mass flow rates etc. The CFD defines the significance of inlet flow rates to analyse the effectiveness of heat exchanger. Higher temperature difference and mass flow rates makes the counter flow heat exchanger pre-eminent among other heat exchanger. Heat exchangers are one of the mostly used equipment in the process industries. Heat exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involve cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purpose. Process fluids, usually are heated or cooled before the process or undergo a phase change. Foli et al. (2011) made two different approaches for the advancement of micro heat exchanger. Considering the geometric parameter and flow condition the two approaches would be like CFD analysis for the optimization of different inlet parameters and other one would be supposedly combination of genetic algorithm and CFD for development in heat exchanger. Kumar et al. (2012) evaluated the thermal performance using Nano fluids instead of using the water as likely to be passive method. They got a better conductivity and heat transfer coefficient rather than the liquid water which provides them the better simulation results for heat transfer however for a two phase model it requires some modification. Das et al. (2012) reviewed the multi stream plate fin heat exchanger. As it is very compact and has a capability to handle different fluid streams well enough than the regular used two-stream heat exchanger. As they stated there is no such universally accepted thermal design for heat exchanger but with the help of differential and numerical analysis & by some area manipulation the heat exchanger effectiveness would be enhanced up to certain extend. Raj et al. (2013) analyses the absorber tube in a parabolic solar collector using some CFD code Ansys CFX 12.0. As the paper suggest efficiency directly depends on the overall surface area. Here the numerical analysis of the absorber using the inserts is validated with the experimental results using water as a working fluid. Peigne et al. (2013) experiment using three fluids for the heat exchanger. Using the flue gas, ventilation air and combustion air the outlet temperature of the specific geometry studied. By computational analysis it shows that if there is any insulation required for the heat exchanger at the outer coating or not. Mohanty et al. (2014) prepares a double pipe heat exchanger using CFD and for checking the performance of heat exchanger ANSYS 14 used. The recent most adapted technique of twisted tape used at the regular spacing, in the middle or at the outer surface for performance evaluation and shows graphically that how the performance enhances at the expense of pressure drop. Thundil et al. (2014) prepared a 3D model for helical coil tube heat exchanger for analysis and optimization of such parameters like temperature and flow rates for various coil pitch. By solving the proper governing equation and boundary condition they have showed to overcome from the flow deviation and avoided the turbulence in the complex flow domain. Kumr et al. (2015) performed the CFD analysis showing that single phase flow can be enhanced by different techniques. Considering two different flow configurations that is parallel and counter flow the experiments were conducted as later compared with both the flow configurations. Aneesh et al. (2016) shows the effect of operating condition with the analysis of thermo-hydraulic properties with helium as a working fluid and alloy as a solid substrate. Performance is predicted for the printed circuit heat exchanger altering the heat transfer density and pressure drop. Nagarseth et al. (2017) gives a better conception of counter flow water tube heat exchanger with CFD analysis. A discrete phase modelling at different zones along the length has been done. MATLAB used to simulate and CFD used for validation of the result with the experimental results. Many researchers have also carried out multi-objective optimization study on several thermal/heat energy converting devices viz. Brayton heat engine, Stirling heat engine, thermoelectric devices etc. [11-35]. The work done on these thermal energy converting devices can also verified with the application of Computational fluid dynamics.

II. GEOMETRY AND MESH GENERATION

The geometry of the heat exchanger in the present study was created in ANSYS 14 as shown in the fig 1. The independent grid test was carried out for the mesh containing 96833, 151962, 200950 and 345826 cells. It was found that the results were same for the geometry containing 200950 and 345826 cells. Therefore, the geometry with 200950 cells was considered in the present study for flow simulation.
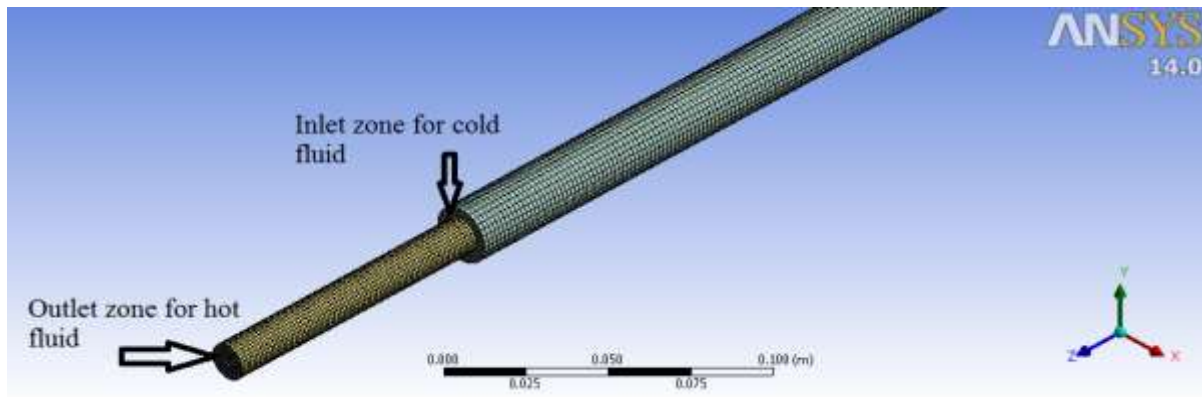

Fig. 1 Schematic diagram of counter flow heat exchanger

III GENERAL GOVERNING EQUATIONS

The continuity equation and the Navier-Stokes momentum equation govern the flow of the fluid in the double tubes.

Continuity equation:

$$\nabla.(\propto_t \rho_t \upsilon_t \upsilon_t)=0 \tag{1}$$

Momentum equation:

$$\nabla.(\propto_t \rho_t \upsilon_t \upsilon_t)=-\propto_f \nabla P+\nabla\tau_f \propto_f \rho_f g+K\_sf(\upsilon_s-\upsilon_f)+C\_vm \propto_f \rho_f g(\upsilon_s \nabla\upsilon_s-\upsilon_f \nabla\upsilon_f)+C\_L \propto_s \rho_f(\upsilon_f-\upsilon_s)\times(\nabla\upsilon_f) \tag{2}$$

The energy balance equation is applied at both the hot and cold inlet.

$$(mc*cpc)(dT\_cout)/dt=(T\_cin*C\_pc*\dot{m}\_h)-(T\_cout*C\_pc*\dot{m}\_c)+(h*A*(T\_hout-T\_cout)) \tag{3}$$

$$(mh*cph)(dT\_hout)/dt=(T\_hin*C\_ph*\dot{m}\_h)-(T\_hout*C\_ph*\dot{m}\_h)+(h*A*(T\_hout-T\_cout) \tag{4}$$

IV Boundary condition

In order to simulate the conditions observed in the experiments, the fully developed parabolic velocity profile needs to be obtained before the fluid enters the tube. The parabolic velocity profile in a laminar flow and the empirical power-law equation in a turbulent flow in the channel with a circular cross-section are obtained and verified [10]. To make the inlet velocity profile to be similar as in the following literature [10], some dynamic boundary conditions are applied. Aluminum is used as a material of construction and water liquid for fluid phase. Using viscous k-ε RNG model turning the energy equation on the iterations continued.

V RESULTS AND DISCUSSIONS

The results for temperature contour of counter flow heat exchanger is shown in the fig. 2 below. As discussed earlier from the first principle, the open loop temperature profile obtained by fixing the cold inlet temperature at 293K and hot inlet temperature at 343K. Fig.2 shown below in the graph shows the open loop temperature profile. The net average temperature obtained after steady state condition is 323.80K. The fig. 3 and fig. 4 shows the temperature variation with respect to the length of the heat exchanger. Fig. 3 shows the cold inlet fluid temperature change across the heat exchanger and fig. 4 shows the hot inlet fluid temperature change across the heat exchanger.
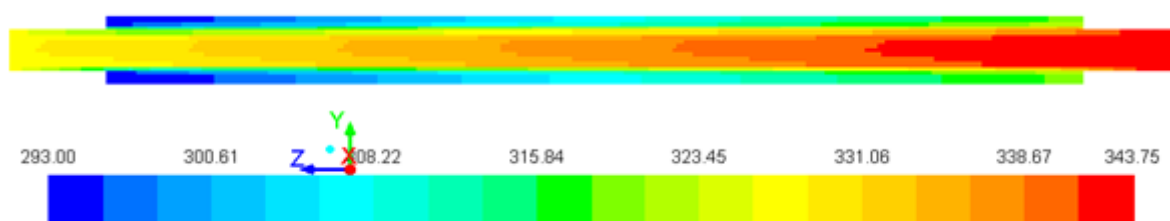
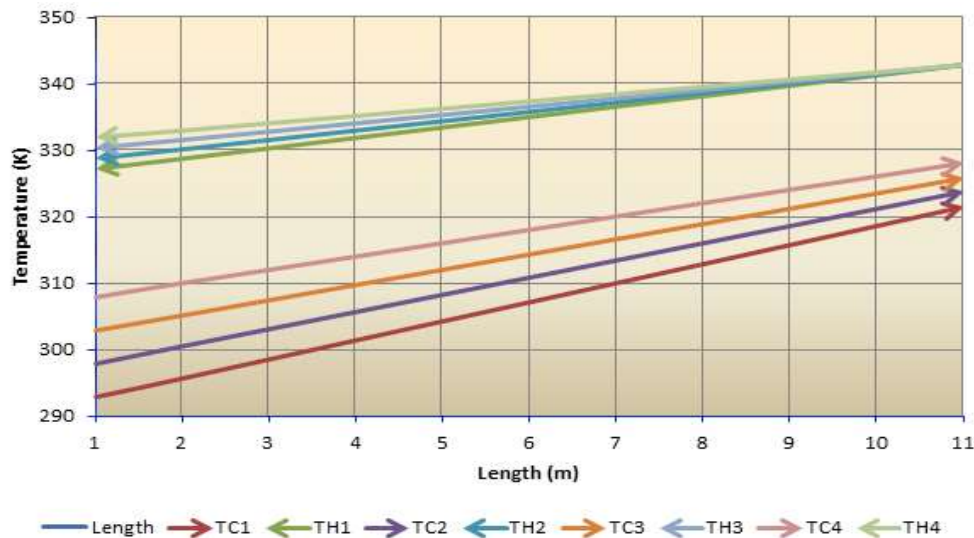**Fig. 2 Temperature contour of counter flow heat exchanger**

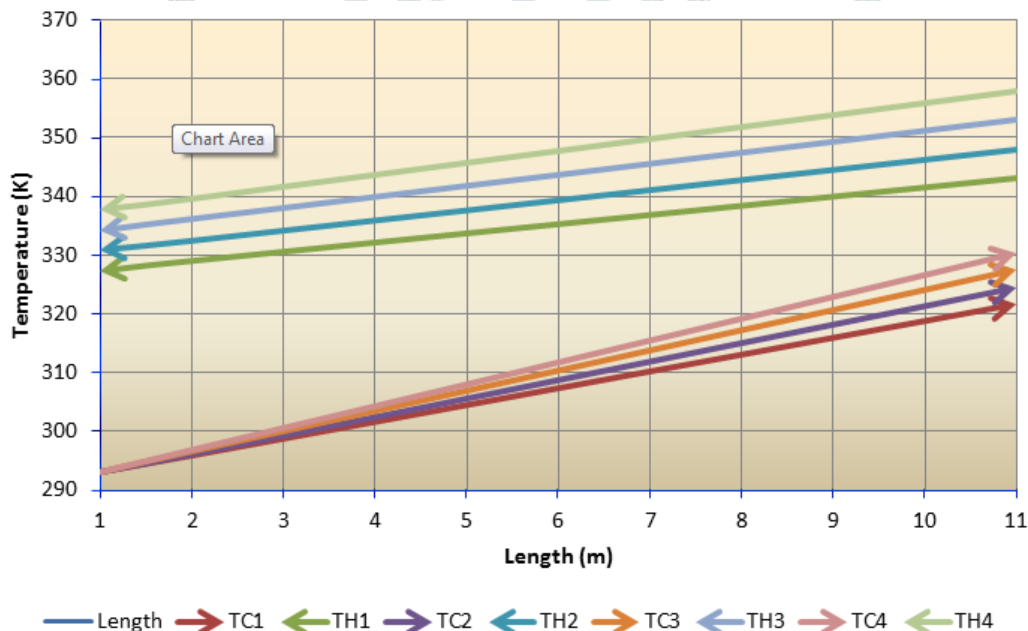**Fig. 3 Cold inlet fluid temperature change**

Fig. 4 Hot inlet fluid temperature change

VI CONCLUSION

The simulation results are depicted are matched with the previous literature [10] and with some analytic and mathematical approach at the boundary condition leads to the better heat transfer. The heat transfer and flow distribution is discussed in detail and proposed model is compared with increased temperature of following model [10]. The model predicts the heat transfer with an average error of less than 2%. Thus the model can be improved. The assumption worked well in this geometry and meshing expects the outlet and inlet region where rapid mixing and change in flow direction takes place We had observed a considerable amount of deviation from the actual value of the followed literature to the value that is obtained in the simulation. This project has further developments like considering different types of flows. And also here we have considered that the radiation and convection losses as zero where as in practical situations they will exist so this project can be further extended in that path. That could have profound effects on the performance of the counter flow heat exchanger. From the pressure and temperature contours it was found that along the outer side of the pipes the velocity and pressure values were higher in comparison to the inner values.

REFERENCES

[1]   Mohammed, H. A., Bhaskaran, G., Shuaib, N. H., & Saidur, R. 2011. Heat transfer and fluid flow characteristics in microchannels heat exchanger using nanofluids: a review. Renewable and Sustainable Energy Reviews, 15(3), 1502-1512.

[2] Kamyar, A., Saidur, R., & Hasanuzzaman, M. 2012. Application of computational fluid dynamics (CFD) for nanofluids. International Journal of Heat and Mass Transfer, 55(15-16), 4104-4115.

[3] Das, P. K., & Ghosh, I. 2012. Thermal design of multistream plate fin heat exchangers—a state-of-the-art review. Heat Transfer Engineering, 33(4-5), 284-300.

[4] Raj, R. T. K., Srinivas, T., Natarajan, M., Kumar, K. A., Chengappa, A., & Deoras, A. 2013. Experimental and numerical analysis using CFD technique of the performance of the absorber tube of a solar parabolic trough collector with and without insertion. In Int conference on Energy Efficient Technologies for Sustainability (ICEETS), IEEE, 550-556.

[5] Peigné, P., Inard, C., & Druette, L. 2013. Ventilation heat recovery from wood-burning domestic flues. A Theoretical Analysis Based on a Triple Concentric Tube Heat Exchanger. Energies, 6(1), 351-373.

[6] Mohanty, R. L., Bashyam, S., & Das, D. 2014. Numerical analysis of double pipe heat exchanger using heat transfer augmentation techniques. International Journal of Plastics Technology, 18(3), 337-348.

[7] Raj, R. T. K., Manoj Kumar, S., Aby Mathew, C., & Elango, T. 2014. Numerical analysis of helically coiled heat exchanger using cfd technique. ARPNJEAS, 9(3), 300-307.

[8] Sivakumar, K., & Rajan, K. 2015. Performance analysis of heat transfer and effectiveness on laminar flow with effect of various flow rates. International Journal of Chem Tech Research, 7(6), 2580-2587.

[9] Aneesh, A. M., Sharma, A., Srivastava, A., Vyas, K. N., & Chaudhuri, P. 2016. Thermal-hydraulic characteristics and performance of 3D straight channel based printed circuit heat exchanger. Applied Thermal Engineering, 98, 474-482.

[10] Nagarsheth, S. H., Bhatt, D. S., & Barve, J. J. 2017. Temperature profile modelling, simulation and validation for a counter flow water tube in tube heat exchanger. In Control Conference (ICC), 2017 Indian, IEEE, 163-169.

[11] Arora, R., Kaushik, S.C. and Kumar, R. 2015. Multi-objective optimization of an irreversible regenerative Brayton cycle using genetic algorithm. In Int Conf on Futuristic Trends on Computational Analysis and Knowledge Management (ABLAZE), IEEE, 340-346 (2015). DOI: 10.1109/ABLAZE.2015.7155017

[12] Arora, R., Kaushik, S.C. and Kumar, R. 2016. Multi-objective thermodynamic optimization of solar parabolic dish Stirling heat engine with regenerative losses using NSGA-II and decision making. Applied Solar Energy, 52 (4): 295-304.

[13] Arora, R., Kaushik, S.C., Kumar, R. and Arora, R. 2016. Multi-objective thermo-economic optimization of solar parabolic dish Stirling heat engine with regenerative losses using NSGA-II and decision making. International Journal of Electrical Power & Energy Systems, 74: 25-35.

[14] Arora, R., Kaushik, S.C. and Kumar, R. 2015. Multi-objective optimization of solar powered Ericsson cycle using genetic algorithm and fuzzy decision making. In 2015 Int Conf on Advances in Computer Engineering and Applications(ICACEA), IEEE, 553-558 (2015). DOI:10.1109/ICACEA.2015.7164754

[15] Arora, R., Kaushik, S.C. and Kumar, R. 2017. Multi-objective thermodynamic optimisation of solar parabolic dish Stirling heat engine using NSGA-II and decision making. International Journal of Renewable Energy Technology, 8 (1): 64-92.

[16] Kumar, Rajesh, Kaushik, S.C. and Kumar, Raj. 2015. Performance Analysis of an Irreversible Regenerative Brayton Cycle based on Ecological Optimization Criterion. International Journal of Thermal and Environment Engineering, 9(1): 25-32.

[17] Kumar, R., Kaushik, S.C. and Kumar, R. 2013. Efficient power of Brayton heat engine with friction. International Journal of Engineering Research and Technology, 6(5): 643-650.

[18] Kumar, R., Kaushik, S.C. and Kumar, R. 2015. Power optimization of an Irreversible regenerative Brayton Cycle using isothermal heat addition. Journal of Thermal Engineering, 1(4): 279-286.

[19] Kaushik, S.C., Kuamar, R. and Arora, R. 2016. Thermo-economic optimization and parameteric study of an irreversible Brayton heat engine cycle. Journal of Thermal Engineering, 2(4): 861-870.

[20] Arora, R., Kaushik, S.C. and Kumar, R. 2015. Performance optimization of Brayton heat engine at maximum efficient power using temp. dependent specific heat of working fluid. Journal of Thermal Engineering, 1(2):345-354.

[21]   Arora, R., Kaushik, S.C., Kumar, Raj and Arora, R. 2016. Soft computing based multi-objective optimization of Brayton cycle power plant with isothermal heat addition using evolutionary algorithm and decision making. Applied Soft Computing, 46:267-283.

[22]   Kumar, R., Kaushik, S.C., Kumar, Raj and Hans, R. 2016. Multi-objective thermodynamic optimization of irreversible regenerative Brayton cycle using evolutionary algorithm and decision making. Ain Shams Engineering Journal, 7(2): 741-753.

[23]   Lal, S., Kaushik, S. C. and Hans, R. 2016. Experimental investigation and CFD simulation studies of a laboratory scale solar chimney for power generation. Sustainable Energy Technologies and Assessments, 13: 13-22.

[24]   Hans, R., Kaushik, S. C., & Manikandan, S. (2015, February). Multi-objective optimization of two stage series connected thermo-electric generator using genetic algorithm. In Futuristic Trends on Computational Analysis and Knowledge Management (ABLAZE), 2015 International Conference on (pp. 165-169). IEEE.

[25]   Hans, R., Kaushik, S. C., & Manikandan, S. (2015, March). Multi-objective optimization of thermo-electric heat pump using genetic algorithm and fuzzy Bellman-Zadeh decision making. In Computer Engineering and Applications (ICACEA), 2015 International Conference on Advances in (pp. 548-552). IEEE.

[26]   Arora, R., Kaushik, S.C. and Arora, R. 2015. Multi-objective and multi-parameter optimization of two-stage thermoelectric generator in electrically series and parallel configurations through NSGA-II. Energy, 91: 242-254.

[27]   Arora, R., Kaushik, S.C. and Arora, R. 2016. Thermodynamic modeling and multi-objective optimization of two-stage thermoelectric generator in electrically series and parallel configurations. Applied Thermal Engineering, 25(103): 1312-1323.

[28]   Arora, R. and Arora, R. 2018. Multiobjective optimization and analytical comparison of single- and 2-stage (series/parallel) thermoelectric heat pumps. International Journal of Energy Research, 1-19. https//doi.org/10.1002/er.3988

[29]   Kaushik, S.C., Manikandan, S. and Hans, R. 2015. Energy and exergy analysis of thermoelectric heat pump system. International Journal of Heat and Mass Transfer, 86: 843-852.

[30]   Kaushik, S. C., Manikandan, S., & Hans, R. 2015. Thermodynamic modelling of thermoelectric generator systems. Heat Pipe Science and Technology, An International Journal, 6 (3-4).

[31]   Hans, R., Manikandan, S. and Kaushik, S.C. 2015. Performance optimization of two-stage exoreversible thermoelectric converter in electrically series and parallel configuration. Journal of Electronic Materials, 44: 3571-3580.

[32]   Hans, R., Kaushik, S.C. and Manikandan, S. 2016. Performance optimization of two-stage exoreversible thermoelectric heat pump in electrically series, parallel and isolated modes. International Journal of Energy Technology and Policy, 12(4): 313-332.

[33]   Kaushik, S. C., Hans, R. and Manikandan, S. 2016. Theoretical and experimental investigations on solar photovoltaic driven thermoelectric cooler system for cold storage application. International Journal of Environmental Science and Development, 7(8): 615.

[34]   Hans, R., Kaushik, S. C. and Manikandan, S. 2016. Experimental study and analysis on novel thermo-electric cooler driven by solar photovoltaic system. Applied Solar Energy, 52(3): 205-210

[35]   Kaushik, S.C., Manikandan, S. and Hans, R. 2016. Energy and exergy analysis of annular thermoelectric heat pump. Journal of Electronic Materials, 45(7): 3400-3409.