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

1 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 signification of second law of thermodynamics. Different computational analysis has been adopted to analyse and configure the heat exchanger with some alteration 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 prominent 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. Foll 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 like 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. Kumar 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.

![Schematic diagram of counter flow heat exchanger](image)

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:

$$\nabla . (\rho_t \mathbf{u}_t) = 0$$

Momentum equation:

$$\nabla . (\rho_t \mathbf{u}_t) = -\rho_f \nabla P + \nabla \tau_f + \rho_f \mathbf{g} + K_s f (v_s - v_f) + C_{vm} \rho_f \mathbf{g} (v_s \nabla v_s - v_f \nabla v_f) + C_L \mathbf{u}_s \rho_f (v_f - v_s) \times (\nabla v_f)$$

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

$$\frac{mc*cpc}{dt} = (T_{cin}*C_pc*\dot{m}_h) - (T_{cout}*C_pc*\dot{m}_c) + (h*A*(T_{hout} - T_{cout}))$$

$$\frac{mh*cph}{dt} = (T_{hin}*C_ph*\dot{m}_h) - (T_{hout}*C_ph*\dot{m}_h) + (h*A*(T_{hout} - T_{cout}))$$

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.
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


