Performance prediction of Counter flow Heat Exchanger by using CFD technique

[1]Shuvam Mohanty, [2]Shofique Uddin Ahmed
[1]M.Tech Student, [2]M.Tech Student
[1]Department of Mechanical Engineering, Amity University, Gurgaon, 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 [8] 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 aluminum 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.**

Index Terms—**3D CFD modeling, RNG K-epsilon model, Temperature distribution, velocity distribution, pressure drop.**
_____

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 modeled 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. 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. 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. 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 modeling 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. 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. 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.

II. COMPUTATIONAL DESIGN MODEL

Computational Fluid Dynamics, abbreviated as CFD, uses different numerical methods and a number of computerized mathematical models in order to solve and analyses problems that involve the flow of fluids. The calculations required simulating the interaction of fluids with surfaces defined by boundary conditions, and initial conditions are done by the ANSYS Fluent v14.0. The Navier stokes equations form the basis of all CFD problems. Two equation models are used for the simulations, and different models are discussed below. The continuity equation and the Navier-Stokes momentum equation govern the flow of the fluid in the counter flow heat exchanger.

Continuity equation:

$$\nabla.(\propto_t \rho_t v_t v_t) = 0$$

Momentum equation:

$$\nabla.(\propto_t \rho_t v_t v_t) = -\propto_f \nabla P + \nabla \tau_f \propto_f \rho_f g + K_{sf}(v_s - v_f) + C_{vm} \propto_f \rho_f g(v_s \nabla v_s - v_f \nabla v_f) + C_L \propto_s \rho_f(v_f - v_s) \times (\nabla v_f)$$

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

$$(\dot{m}_c \times C_{pc})\frac{dT_{cout}}{dt} = (T_{cin} * C_{pc} * \dot{m}_h) - (T_{cout} * C_{pc} * \dot{m}_c) + (h * A * (T_{hout} - T_{cout}))$$

$$(\dot{m}_h \times C_{ph})\frac{dT_{hout}}{dt} = (T_{hin} * C_{ph} * \dot{m}_h) - (T_{hout} * C_{ph} * \dot{m}_h) + (h * A * (T_{hout} - T_{cout})$$

Reynolds tensor for fluid while applying RNG $k - \varepsilon$ is

$$\tau_{t,f} = -\frac{2}{3}(\rho_f K_f + \mu_{t,f} \nabla v_f)I + \mu_{t,f}(\nabla v_f + \nabla v_f^{tr}$$

$$\mu_{t,f} = \rho_f C_\mu \frac{k_f^2}{\in_f}$$

With

$$C_\mu = 0.0845$$

The prediction of turbulent kinetic energy and rate of dissipation obtained from the following equation;

$$\nabla.(\alpha_f \rho_f U_f \varepsilon_f) = \nabla.\left(\alpha_f \frac{\mu_{t,f}}{\sigma_\varepsilon} \nabla \varepsilon_f\right) + \alpha_f \frac{\varepsilon_f}{k_f}(C_{1\varepsilon} G_{k,f} - C_{2\varepsilon} \rho_f \varepsilon_f) + \alpha_f \rho_f \prod_{\varepsilon,f}$$

The RNG model constants are
$C_\mu = 0.0845$
$C_{1\varepsilon} = 1.42$
$C_{2\varepsilon} = 1.68$
Wall Prandtl number = 0.85

III. NUMERICAL SOLUTION

A Commercial CFD software has used for the simulation and validation. ANSYS provides everything under one platform i.e. design modeling, meshing, complicated flow simulation. A 3D geometry of counter flow heat exchanger with some modification in the design is generated in the ANSYS Fluent v14 design modular and modified to two computational zones for advance meshing. The governing energy equation and temperature fields are discretized into two different mesh zones and then the subsequent equations are solved by the help of iterative method. For plotting the temperature contours the ANSYS set up has been used.
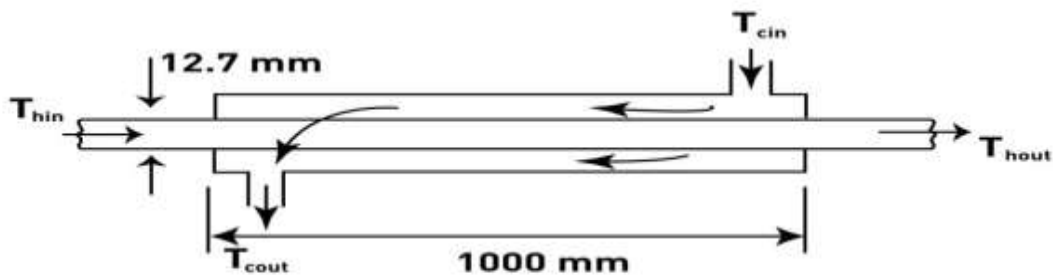
Fig. 1 schematic diagram of counter flow heat exchanger [8]

IV. MESH GENERATION

Initially a relatively coarser mesh is generated with 96833 cells. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured cells (Hexahedral) as much as possible, so further we switched to the medium mesh which generates 151962 cells with 139851 numbers of nodes. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, for the mesh independent model, a fine mesh is generated with 200950 cells. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed. But the fine mesh lies very close to the medium mesh and considering the medium mesh the results for the temperature change comes out very good. So that's why we go with the medium mesh.
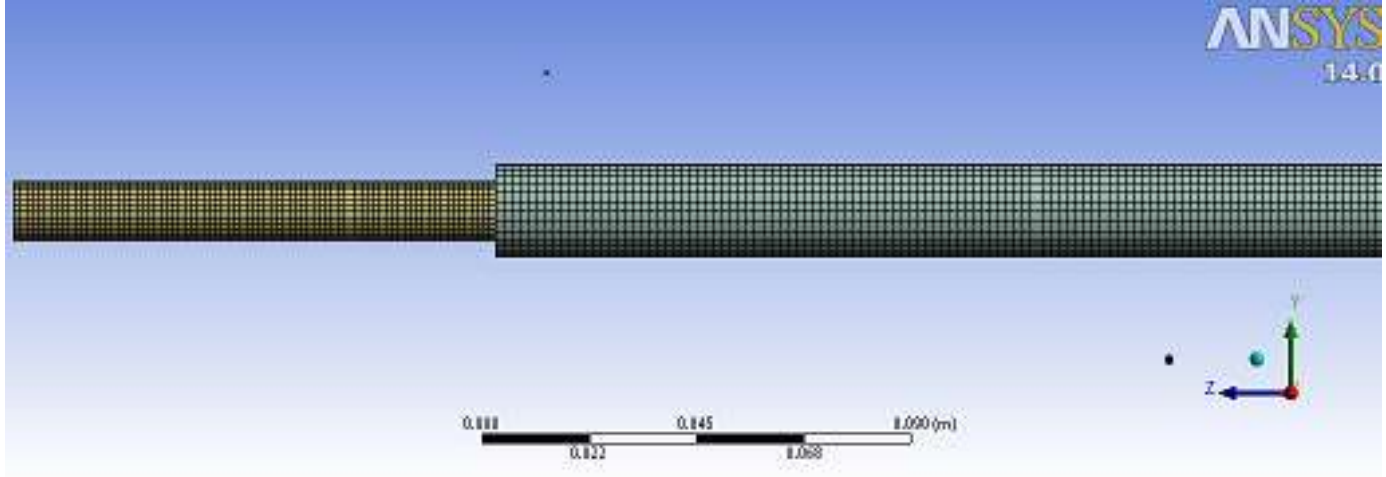

Fig.2 Mesh generation

V. 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 [8]. To make the inlet velocity profile to be similar as in the following literature [8], 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.

VI. RESULTS AND DISCUSSION

The convergence of Simulation is required to get the parameters of the counter flow heat exchanger in outlet. It also gives accurate value of parameters for the requirement of heat transfer rate. Continuity, X-velocity, Y-velocity, Z-velocity, energy, k-epsilon are the part of scaled residual which have to converge in a specific region. For the continuity, X-velocity, Y- velocity, Z-velocity, k-epsilon should be less than 10^{-4} and the energy should be less than 10^{-7}. Values and boundary conditions mentioned above leads to convergence criteria. As discussed earlier from the first principle, the open loop temperature profile obtained by fixing the cold inlet temperature at 298K and hot inlet temperature at 348K. 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.
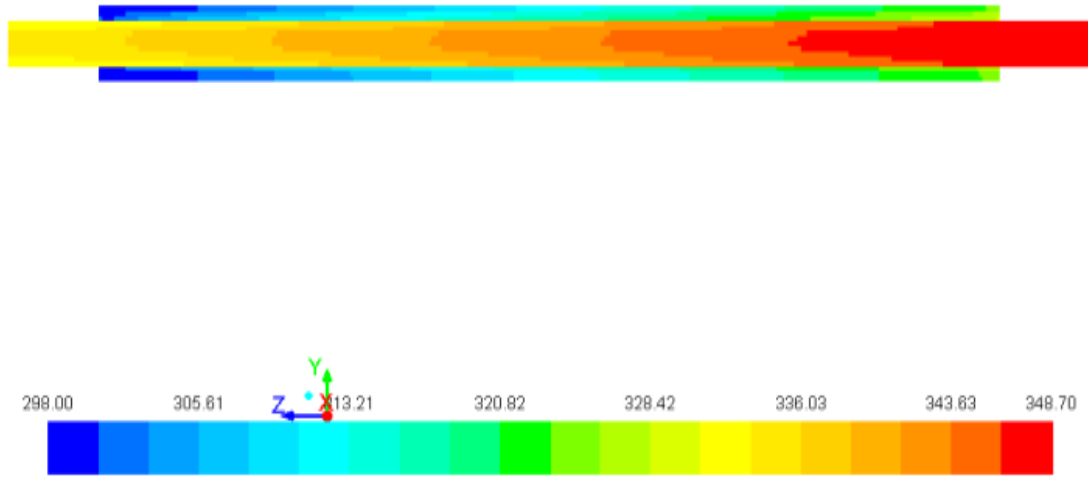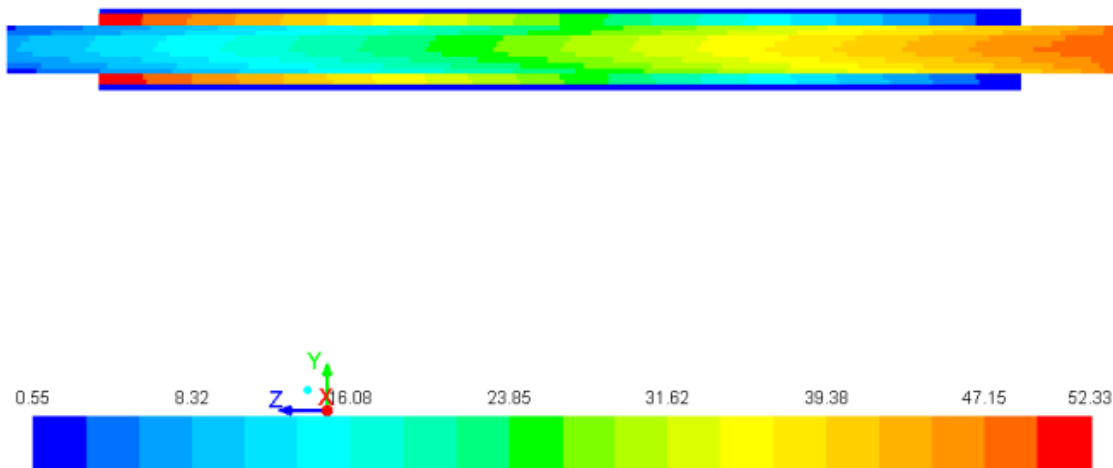
Fig. 3 Temperature Contour (298-348)K

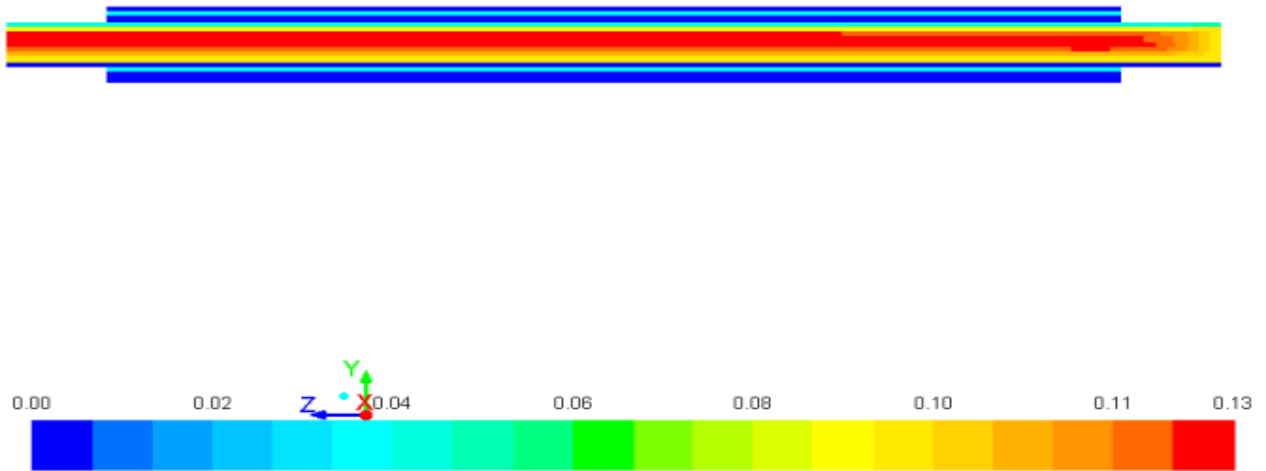**Fig. 4 pressure contour for temperature range (298-348)K**

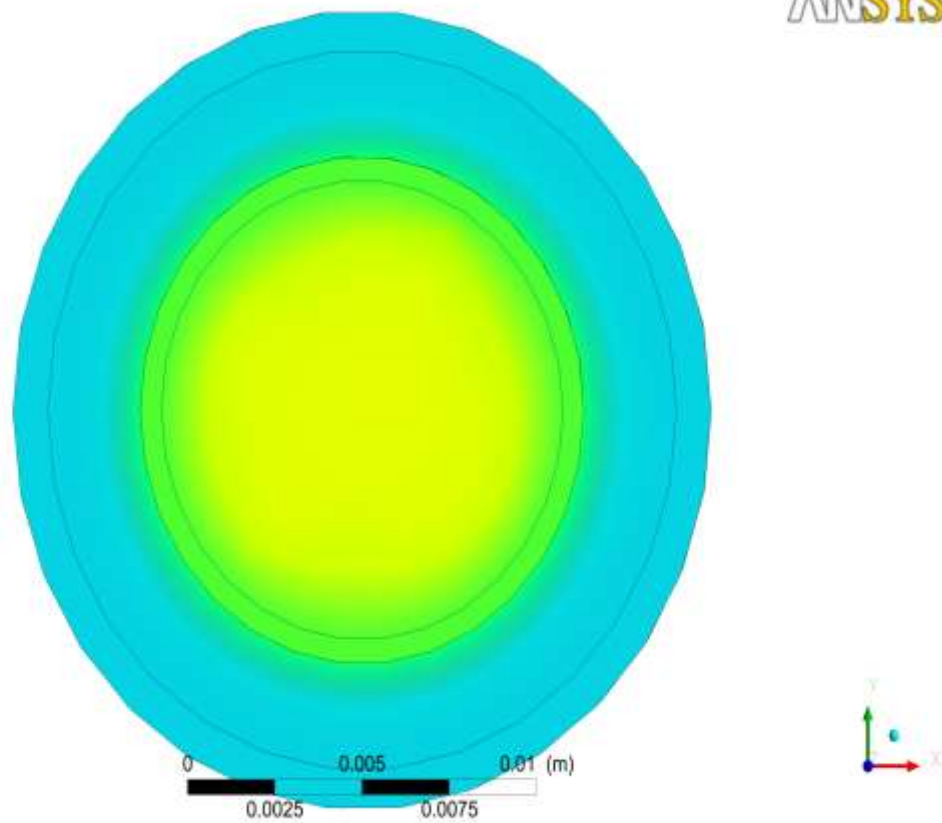**Fig. 5 Velocity contours for temperature (298-348)K**
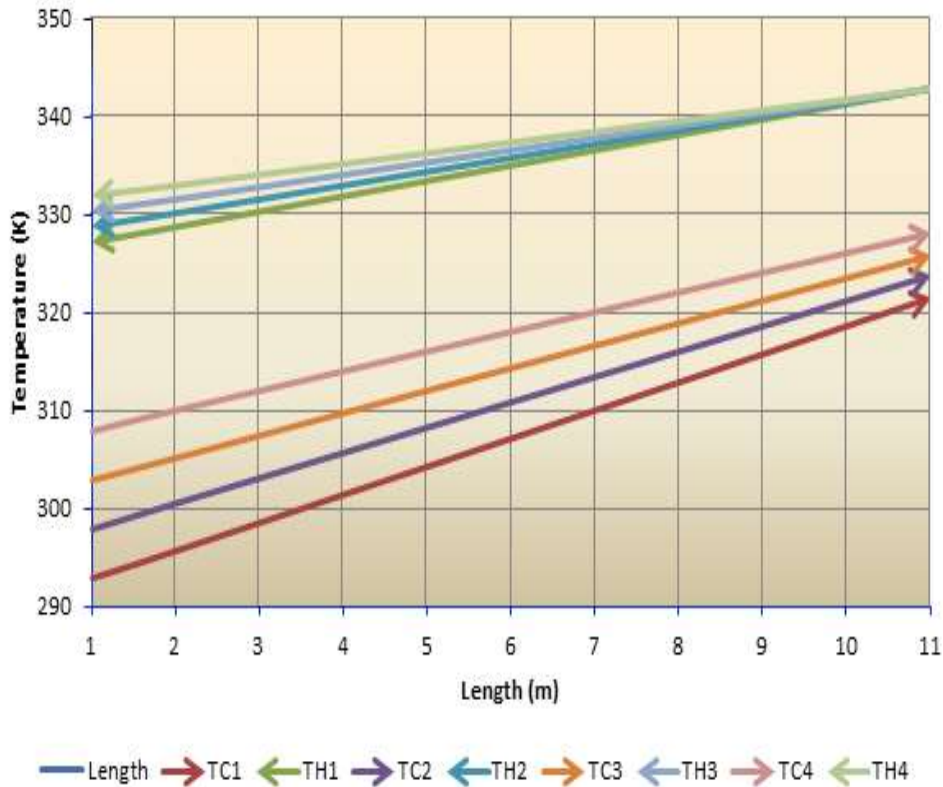

Fig. 6 Temperature profile from the face

Fig. 7 Graphical representation of cold inlet temperature variation

By step changing the input temperature of cold inlet fluid and hot inlet fluid the temperature profile are studied. The results are also validated as shown in the fig.6 below. It also shows the current results are better than the following literature [8]. As the heat transfer increases the LMTD results also affects. The LMTD increases along with that heat transfer.
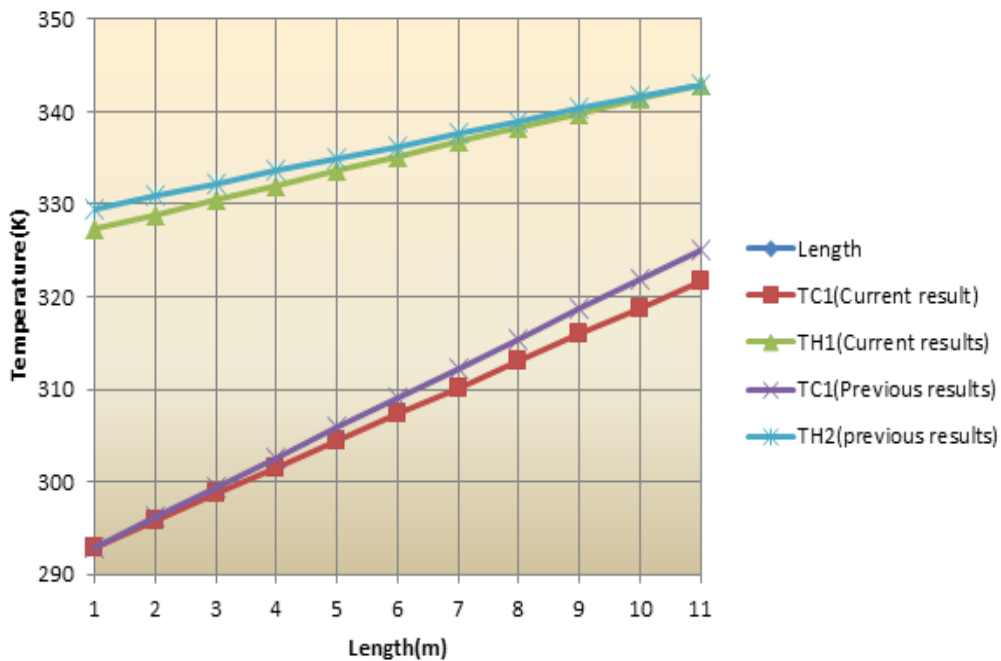

Fig. 8 Validation of result

VII. CONCLUSION

The simulation results are depicted are matched with the previous literature [8] 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 [8]. 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., et al. "Heat transfer and fluid flow characteristics in microchannels heat exchanger using nanofluids: a review." *Renewable and Sustainable Energy Reviews* 15.3 (2011): 1502-1512.

[2] Kamyar, A., R. Saidur, and M. Hasanuzzaman. "Application of computational fluid dynamics (CFD) for nanofluids." *International Journal of Heat and Mass Transfer* 55.15-16 (2012): 4104-4115.

[3] Das, Prasanta Kumar, and Indranil Ghosh. "Thermal design of multistream plate fin heat exchangers—a state-of-the-art review." *Heat Transfer Engineering* 33.4-5 (2012): 284-300.

[4] Raj, R. Thundil Karuppa, et al. "Experimental and numerical analysis using CFD technique of the performance of the absorber tube of a solar parabolic trough collector with and without insertion." *Energy Efficient Technologies for Sustainability (ICEETS), 2013 International Conference on*. IEEE, 2013.

[5] Mohanty, Rajiva Lochan, Sharon Bashyam, and Darpahari Das. "Numerical analysis of double pipe heat exchanger using heat transfer augmentation techniques." *International Journal of Plastics Technology* 18.3 (2014): 337-348.

[6] Raj, R. Thundil Karuppa, et al. "Numerical analysis of helically coiled heat exchanger using cfd technique." *ARPNJEAS* 9.3 (2014): 300-307.

[7] Aneesh, A. M., et al. "Thermal-hydraulic characteristics and performance of 3D straight channel based printed circuit heat exchanger." *Applied Thermal Engineering* 98 (2016): 474-482.

[8] Nagarsheth, Shaival H., Dhruvi S. Bhatt, and Jayesh J. Barve. "Temperature profile modelling, simulation and validation for a counter flow water tube in tube heat exchanger." *Control Conference (ICC), 2017 Indian*. IEEE, 2017.

[9] Peigné, Pierre, Christian Inard, and Lionel Druette. "Ventilation heat recovery from wood-burning domestic flues. A Theoretical Analysis Based on a Triple Concentric Tube Heat Exchanger." *Energies* 6.1 (2013): 351-373.

[10] Sivakumar, K., and K. Rajan. "Performance analysis of heat transfer and effectiveness on laminar flow with effect of various flow rates." *International Journal of Chem Tech Research* 7.6 (2015): 2580-2587.