CDF Terminology for Helical Pipe Tube Segments of Heat Exchanger

Shubham Pathak[1] Prof. Krishanpal Singh Rajpoot[2]
[1]Research Scholar [2]Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]Bhopal Institute Technology and Science (BITS), Bhopal (M.P.), India

Abstract— The view of CFD analysis of 200 mm long helical pipe segment of heat exchanger with five turn. We have check on performance analysis by CFD for inlet, outlet and wall condition and boundary condition with its degree of freedom. Efficiency of heat exchanger and its dimensions are ones of the most important parameters to consider in engineering design. The size of heat exchanger can be more compact by introducing the fins to increase the heat transfer rate between the heat exchanger surface and the surroundings. Different engineering methods are used in heat exchanger design process. The proper correlations or modelling and simulation tools are often applied to receive the general recommendation at early stages of exchanger study. The performance of the fin-tube heat exchanger for different fin thickness is calculated. To give indications about the accuracy of numerical outcome, the most popular correlations are evaluated and results obtained from Ansys FLUENT program are verified. Analyzing the output, it seems that the implementation of the CFD model. My project aims to perform a numerical study of helical coil tube-in-tube heat exchanger with water as both hot and cold fluid. To improve the effectiveness and its efficiency through this analysis.
Keywords: CFD, Helical tube, heat exchange of CFD

I. INTRODUCTION

A heat exchanger is a heat transfer device that exchanges heat between two or more process fluids. Heat exchangers have widespread industrial and domestic applications. Many types of heat exchangers have been developed for use in steam power plants, chemical processing plants, building heat and air conditioning systems, transportation power systems, and refrigeration units.

The actual design of heat exchangers is a complicated problem. It involves more than heat-transfer analysis alone. Cost of fabrication and installation, weight, and size play important roles in the selection of the final design from a total cost of ownership point of view. In many cases, although cost is an important consideration, size and footprint often tend to be the dominant factors in choosing a design.

II. PROBLEM FORMULATION

Heat exchangers are devices which do not require power to operate them, but they need power in order to pump the working fluid in and out of the device's system. The Performance will be enhanced if the power required to pump the fluids is within acceptable limits according to the heat transfer occurring between the fluids, i.e. the effectiveness and efficiency of the heat exchanger. There are three goals that are normally considered in the optimal design of heat exchangers:

1) Minimizing the pressure drop (pumping power),
2) Maximizing the thermal performance and
3) Minimizing the entropy generation (thermodynamic), which are to be achieved in any possible way known or not known? Thus, an optimum method is to be evaluated, for which the performance of heat exchanger is boosted for same pumping power and ideally without any pressure drops.

III. AIM OF THE PRESENT WORK

The design of a helical coil tube in tube heat exchanger has been facing problems because of the lack of experimental data available regarding the behaviour of the fluid in helical coils and also in case of heat transfer data, which is not the case in Shell & Tube Heat Exchanger. For that we are doing numerical analysis was carried out to determine the heat transfer characteristics for a double-pipe helical heat exchanger by varying the different parameters like different temperatures and diameters of pipe and coil and also to determine the fluid flow pattern in helical coiled heat exchanger. The objective of the project is to obtain a better and more quantitative insight into the heat transfer process that occurs when a fluid flows in a helically coiled tube. The study also covered the different types of fluid flow range extending from laminar flow through transition to turbulent flow.

Heat exchanger is built in the ANSYS workbench design module. It is a counter-flow heat exchanger. First, the fluid flow (fluent) module from the workbench is selected. The design modeller opens as a new window as the geometry is double clicked.

To improve the performance of heat exchanger by changing tube geometry. To test the performance using Computational Fluid Dynamics (CFD) using appropriate simulation tools.

IV. SPECIFICATION OF DIMENSION

Diameter of inner inlet = 40mm
Diameter of inner pipe = 45mm
Thickness of pipe = 5 mm
Diameter of outer inlet = 80mm
Length of pipe = 200mm
Pitch of the pipe (p) = 40mm
Turn of pipe (n) = 5
Material = copper/aluminum

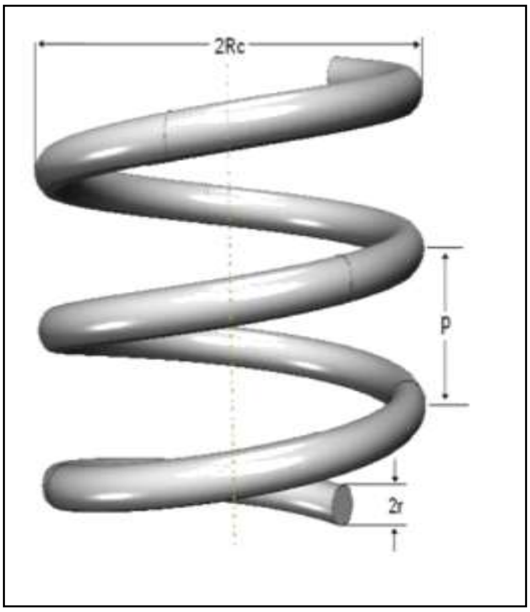

Fig. 1: Dimension

V. SPECIFICATION OF DESIGN

T_i = 310k (hot fluid entrance)
T_i = 298k (cold fluid entrance)
Viscosity = 0.001003 kg/m-s
Ratio of specific heats = 1.4
Pressure = 0.3 Pascal
Density = 1 kg/m3
Thermal conductivity (K) =0.6 W/mK

VI. GEOMETRY OF MODEL

The geometry drawn and import to ANSYS fluent, it's an isomeric view.

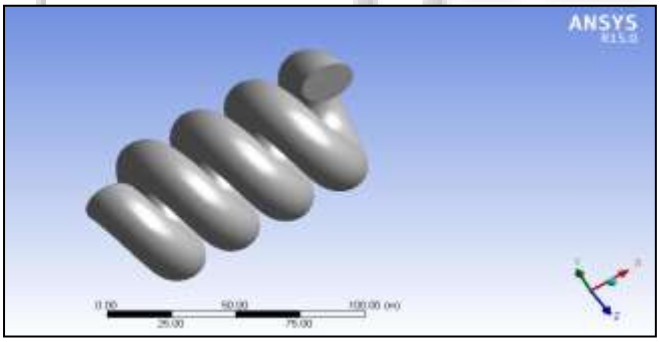

Fig. 5.2: Geometry of helical pipe

VII. MESHING OF MODEL

Meshing is a process to assign to start the CFD analysis with the help direction of inlet; outlet and wall. Heighted zones are direction.

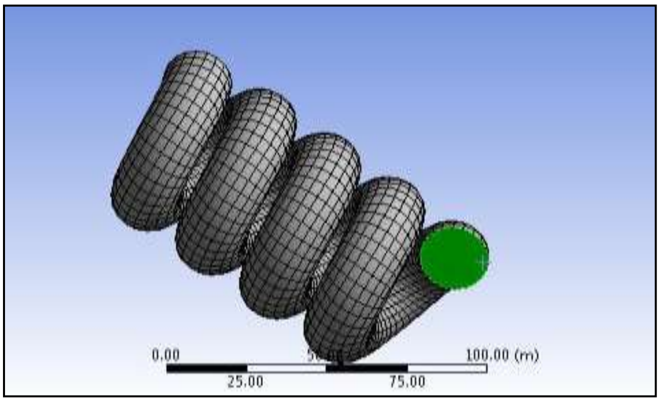

Fig. 5.3: Meshing of model

VIII. MESHING DETAIL

Element Type	Tetrahedron
Nodes	89520
Elements	43170

Table 5.1: Meshing detailing

IX. BOUNDARY CONDITIONS

The inlet and outlet conditions have been taken as velocity inlet and pressure outlet respectively. There will be two inlet and two outlet for the flow as this is counter flow. There is a pipe that separates the two flows is made by copper. Copper was taken as the base metal because of its high value of thermal conductivity. The details of all boundary conditions are listed below. Fluid in the inner pipe was taken as hot fluid with 310K and inn the outer fluid cold fluid at 298K. The calculation is done for different D/d ratio with varying inlet velocity of hot fluid and the inlet velocity for the cold fluid was taken constant. The velocity of hot fluid ranges from 0.8-1.95m/s.

X. RESULT & DISCUSSION

The numerical simulation is done by ANSYS FLUENT software to show both the flow field and heat transfer of the present models. Many cases are studied. Three cases are discussed in the following sections. Same boundary conditions are used in the three cases, which are velocity, inlet temperature and flow rate also respectively.

The various result contours are as follows such as velocity contour and static temperature contour variation.
The turbulent intensity of 5% of inlet and outlet.

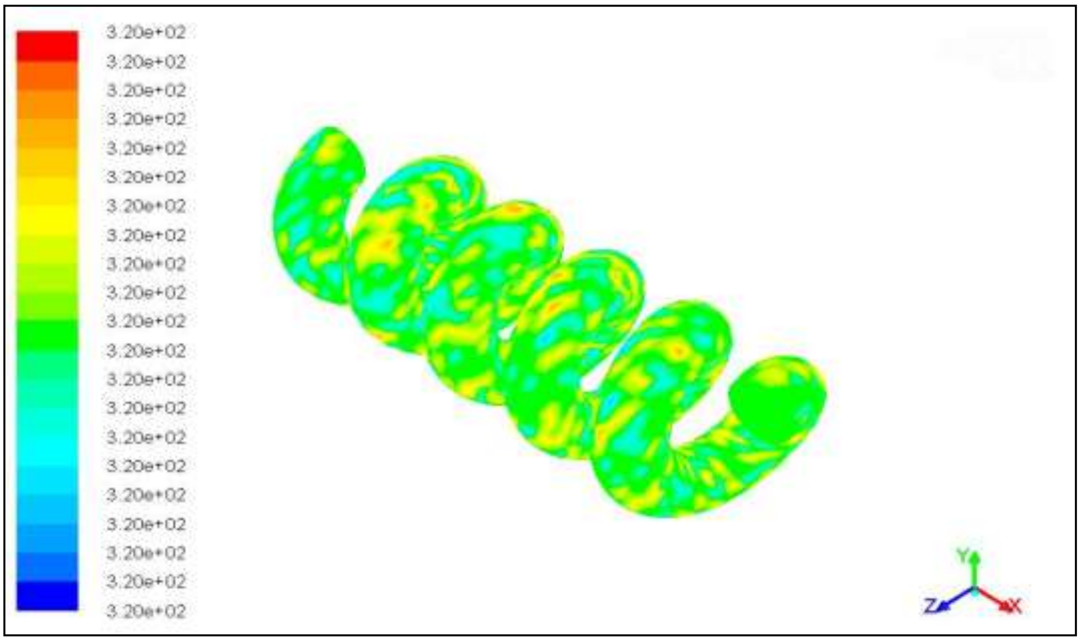

Fig. 5.4: Velocity contour

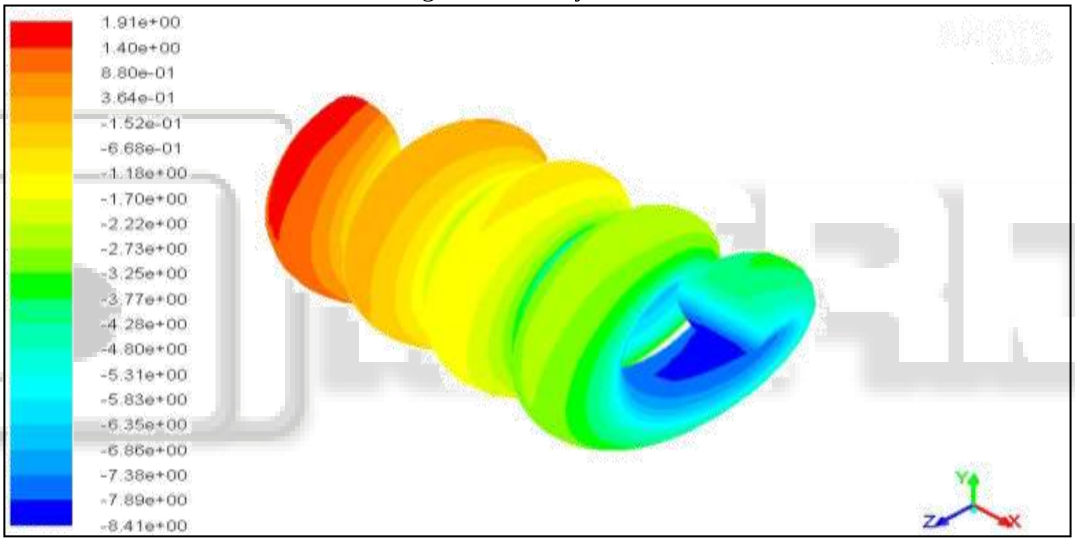

Fig. 5.5: Temperature contour

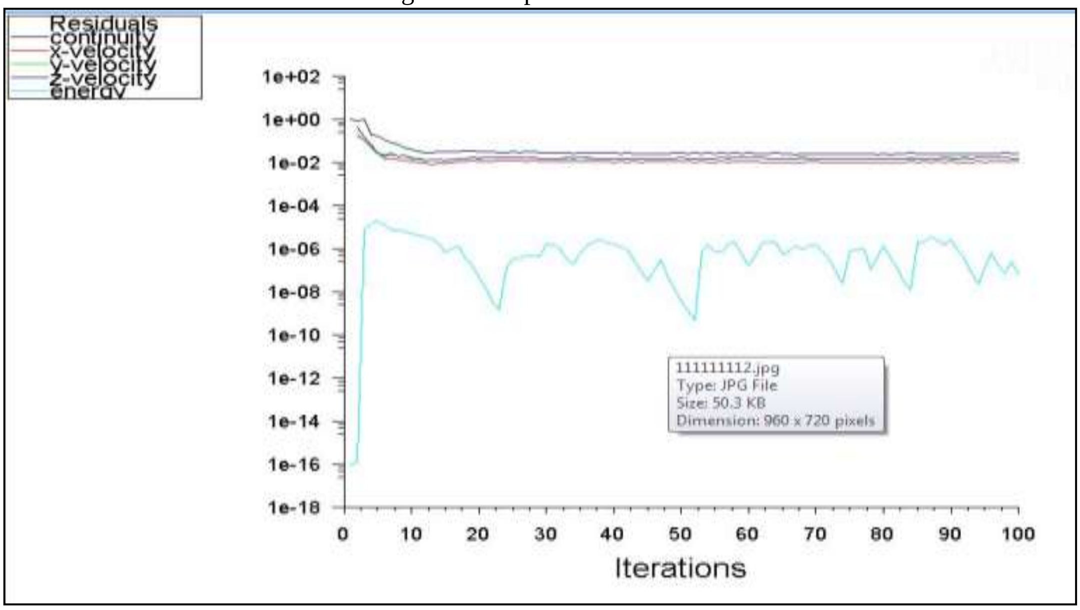

Fig. 5.6: Result graph

XI. Conclusions

The main objective of this research is to determine numerically the performance of the heat transfer process in a single row fin-tube cross flow heat exchanger for different fin configurations. The most popular correlations are applied for heat transfer evaluation. For Briggs and Young correlation, the heat transfer decreases with the turns of pipe increase. The opposite results are seen for the other correlations. The heat transfer is also analyzed by means of numerical computation. The results are verified with the known correlations for various turns of constant thickness. Analyzing the output received from numerical calculations with those gathered from correlations, it seems that the differences are within the standard deviation and numerical techniques can predict heat transfer coefficients with acceptable accuracy. The use of the CFD model offers particular benefits especially when minor modification are applied to the fin surface for which the correlation equations are not defined, for instance fin thickness modification. However, comparative analyses are still required and the numerical model should be examined, verified with proper correlations or experimental values.

Reference

[1] Avinash P. Shinde1, Mahadev L. Shinde2, Sagar S. Yadav3, "Performance Analysis of Heat Exchanger using CFD", IJIRSET, Vol. 7, Issue 3, March 2018.

[2] M.D.Rajkamal, M. Mani Bharathi, Shams Hari Prasad M, Santhosh Sivan.M," Thermal analysis of shell and tube heat exchanger",IJPAM, Volume 119 No. 12 2018, 14299-14306.

[3] R. J. Yadav and A. S. Padalkar," CFD Analysis for Heat Transfer Enhancement inside a Circular Tube with Half-Length Upstream and Half-Length Downstream Twisted Tape", Journal of Thermodynamics, Volume 2012,, 12 pages.

[4] Pramod Deshmukh, Vikram D Patil," cfd analysis of heat transfer in helical coil tube in tube heat exchanger", ijiert, volume 3, issue 1, jan.-2016.

[5] Bharat Bhushan Verma, Saurabh Kumar," CFD Analysis and Optimization of Heat Transfer in Double Pipe Heat Exchanger with Helical-Tap Inserts at Annulus of Inner Pipe", IOSR-JMCE, Volume 13, Issue 3 Ver. VII (May- Jun. 2016), PP 17-22.

[6] Mohammed Irshad, Mohammed Kaushar, G. Rajmohan," Design and CFD Analysis of Shell and Tube Heat Exchanger", IJESC, 2017 Volume 7 Issue No.4.

[7] Umang K Patel, Prof. Krunal Patel," CFD ANALYSIS HELICAL COIL HEAT EXCHANGER", IJARIIE, Vol-3 Issue-2 2017.

[8] K.Ravikumar, Ch.Naga Raju, Meera Saheb," CFD Analysis of a Cross-flow Heat Exchanger with Different fin thickness", International Journal of Dynamics of Fluids. ISSN 0973-1784 Volume 13, Number 2 (2017), pp. 345-362.

[9] Mohamed f. Al-dawody, dhafer a. Hamzah & mushtaq faisal al-mensory," experimental and cfd analysis of heat transfer in a Double pipe helical coil heat exchanger", (ijmperd),Issn (p): 2249-6890; issn (e): 2249-8001,Vol. 8, issue 4, aug 2018,pp. 959-968.

[10] Iraida Krutova and Yakov Zolotonosov," Solution of conjugate problem in a conical coil heat exchanger", STCCE-2020, Materials Science and Engineering (2020), pp.1-7.

[11] M. Thirumarimurugan, T.Kannadasan and E.Ramasamy, Performance Analysis Of Shell And Tube Heat Exchanger Using Miscible System, American Journal Of Applied Sciences 5 (5): 548-552, 2008.

[12] K. Sudhakara Rao, Analysis Of Flow Mal distribution In Tubular Heat Exchangers By Fluent, National Institute Of Technology Rourkela ,2007.

[13] M.R. Salimpour, Heat Transfer Coefficients of Shell and Coiled Tube Heat Exchangers, Isfahan University Of Technology, Iran, 2008.

[14] Yusuf Ali Kara, Ozbilen Gurarasa, A Computer Program For Designing Of Shell-And-Tube Heat Exchangers, Applied Thermal Engineering University Of Ataturk, Turkey, 2004.

[15] Usman Ur Rehman, Heat Transfer Optimization of Shell-And-Tube Heat Exchanger through CFD Studies, Chalmers University of Technology, 2011.

[16] Huadong Li And Volker Kottke, Effect Of The Leakage On Pressure Drop And Local Heat Transfer In Shell-And-Tube Heat Exchangers For Staggered Tube Arrangement, Inl. J. Heat Mass Transfer, Elsevier Science Ltd., 1998.

[17] Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao, A Design And Rating Method For Shell-And-Tube Heat Exchangers With Helical Baffles, Journal Of Heat Transfer, May 2010.

[18] Huadong Li And Volker Kottke, Effect Of Baffle Spacing On Pressure Drop And Local Heat Transfer In Shell-And-Tube Heat Exchangers For Staggered Tube Arrangement, Inl. J. Heat Mass Transfer, Elsevier Science Ltd., 1998.

[19] Muhammad Mahmood Aslam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, Sarfaraz Khan5, CFD Applications In Various Heat Exchangers Design: A Review, Department Of Mechanical Engineering, University Of Engineering & Technology, Applied Thermal Engineering, 2011.

[20] Philip J Stop ford, Recent Application of CFD Modeling in Power Generation Metal and Process Industries, Second International Conference on CFD in Mineral and Process Industries, CSIRO, Melbourne, Australia, 1999.

[21] Khairun Hasmadi Othman, CFD Simulation of Heat Transfer In Shell And Tube Heat Exchanger, University Malaysia Pahang, April 2009.

[22] .https://www.researchgate.net/profile/Bahman_Zohuri/publication/308464270_Heat_Exchanger_Types_and_Classifications/links/5a9851b3aca27214056d479d/HeatExchanger-Types-and-Classifications.pdf.