International journal of Master of Engineering Research and Technology, 2394-6172-(O)
Volume-4, Issue-8, August 2017

Enriched Heat Transfer and Pressure Drop Analysis in Helically Coiled Tube using Different Nanofluid

Hardesh Kumar Chaturvedi
M. Tech. Scholar,
Department of ME,
PCST, Bhopal
hardesh.chaturvedi91@gmail.com

Chhtrpal Temre
Assistant Professor,
Department of ME,
PCST, Bhopal
chhtrpaltemre09@yahoo.com

Amit Kumar
Assistant Professor
Department of ME,
PCST, Bhopal
Amitavit689@gmail.com

ABSTRACT

Heat transfer fluids have many industrial and civil applications, including electronic cooling, transport, energy and air-conditioning system. Traditional fluids such as water, oil and glycols have poor thermal performance due to their low thermal conductivities [1-2]. Research and development has been carried out to improve the thermal performance of fluid. In a computational fluid dynamic, trails have been conducted on the heat transfer and nusselt no. for a helical coil tube heat exchanger. In the CFD investigation, it was assured that the two types of nanofluids were considered with water also to determine overall heat transfer rate and nusselt no., for heat transfer. This investigation was carried out on a different mass flow rate ranging between 0.00418 to 0.0167 Kg/s. When the observations were compared between water and other nanofluids it was concluded that: The CuO as a nano fluid could not transfer the desired heat and so it was not practically prepared and the nano fluid as Fe2O3 is efficient enough to transfer heat but water has a huge complexity in design, whereas the Fe2O3 as a nano fluid have overcome this problem.

Keywords: - ***CFD Analysis, Nusselt No., Fluent, Helical Coil Tube Heat Exchanger, Nusselt No., Overall Heat Transfer Coefficient.***

INTRODUCTION

Heat transfer fluids has been recognized that the suspension of solids in fluids enhances the effective thermal conductivity of the material [1]. Enhancement of thermal conductivity of fluids contributes to improving the efficiency of heat transfer fluids. Furthermore, it is possible to reduce the size of the heat exchange system, which has been limited due to the poor thermal transport property of fluids[3]. In earlier days, micron-sized or larger particles were suspended in fluids and they led to causing the problems, such as the settlement of particles, clogging, abrasion of devices, etc. Manufacture process of TTHC heat exchanger is schematically working process. However, some significant considerations should be attended in this method as described below. It presents a view of a special machine which is extremely employed to trans-form a strait tube into a helical tube in industry. Hence, straight tubes were inserted inside each other before twist operation[5-8]. In this step, inside of the inner and outer tube should be filled up with uncompressible fluid (liquid) in order to prevent unwanted cross section deformations. Here water was employed as an uncompressible fluid for this purpose. Indeed, both inner and outer tubes were filled up with water and the inlets and outlets of heat exchanger were quite closed before coiling operation[12-15].
Rest of this paper is organized as follows in Section 2 discusses about Computational Fluid Dynamic, Section 3 discusses about the proposed model and modified algorithm. Sections 4 describe the experimental result and finally discuss the conclusion & future scope in Section 5.

2. COMPUTATIONAL FLUID DYNAMIC (CFD)

The purpose of this dissertation is to use simulate temperature distribution and wind velocity stream lines in greenhouse dryer heat and validate the simulation with an actual Base paper result. Different solvers and turbulence models are used to try to determine the most accurate CFD method. Computational fluid dynamics (CFD) may be a computer-based simulation methodology for analyzing fluid flow, heat transfer, and connected phenomena like chemical reactions. This thesis uses CFD for analysis of temperature and wind velocity[2-6].

It may be advantageous to use CFD over traditional Base paper result-based analyses, since experiments have a cost directly proportional to the number of configurations desired for testing, unlike with CFD, where massive amounts of results may be created at practically no added expense. During this method, parametric studies to optimize instrumentation are terribly inexpensive with CFD in comparison to Base paper result[8-10].

This section shortly describes the overall concepts and theory associated with using CFD to analyses fluid flow and heat transfer, as relevant to this thesis. It begins with a review of the tools required for carrying out the CFD analyses and the processes needed.

STEPS IN COMPUTATIONAL FLUID DYNAMICS

CFD codes are structure around the numerical algorithms that may take fluid flow issues. So as to produce easy accessibility to their resolution power these codes include sophisticated user interface to input drawback parameters and to examine the results. Thus all codes contain 3 main components[12-14]

- Pre – processor
- Solver and
- Post –processor.

Pre –Processor

Pre – processing consists of the input flow drawback to a CFD programs by means that of an operate-friendly interface and the subsequent transformation of this input into a form appropriate to be used by the solver. The user activities at the pre-processing stage involve[9]:

- Definition of the geometry of the region of interest: the computational domain.
- Grid generation the sub –division of the domain into variety of smaller, non-overlapping sub – domain: a grid (or mesh) of cells (or control volumes or elements).
- Selection of the physical and chemical phenomena that require to be modelled.
- Definition of fluid properties.
- Specification of appropriate boundary conditions at cell that coincides with or touch the boundary.

The solution to a flow drawback (velocity, pressure, temperature etc.) is outlined at nodes within every cell. The accuracy of a CFD resolution is governed by the number of cells within the grid. In general, larger the number of cells better is the solution accuracy. The accuracy of a solution and its cost in terms of necessary computer hardware and calculation mass flow rate inlet and mass flow rate outlet are dependent on the fineness of the grid. Optimal mesh is typically non-uniform: finer in areas where large variations occur from point to point and coarser in regions of relatively little changes. Efforts are beneath way to develop CFD codes with a self -adaptive meshing capability. Ultimately such programs will automatically refine the grid in areas of fast variations. A considerable amount of basic development work still needs to be done before these techniques are robust enough to be incorporated into commercial CFD codes. At present it's still up to the skills of the CFD user to design a grid that's an appropriate compromise between desired accuracy and resolution cost. Over 500th of the mass flow rate inlet and mass flow rate outlet spent in industry on a CFD project is dedicated to the definition of the domain geometry and grid generation[6-7].

Solver

There are three distinct streams of numerical solution techniques: finite difference, finite element and finite volume method. The numerical methodology that kind the premise of the solver performs the subsequent steps[4]:

- Approximation of the unknown flow variables by means of simple functions.
- Discretization by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.
- Solution of the algebraic equations.

The main distinction between the separate streams are associated with the method in which the flow variables are approximated and with the discretization processes.

Post-Processor

- Post-processor is a set of versatile data visualization tools. These include:
- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- View manipulation (translation, rotation, scaling etc.)

- Colour postscript output

3. PROPOSED WORK AND MODELING

Geometry was modelled in ANSYS 15.0 and then generated to ANSYS workbench 15.0 where meshing was done, then the mesh was generated to FLUENT. The boundary conditions, material properties and encompassing properties were set through parameterized case files. FLUENT solves the problem until either the convergence limit is met or the amounts of iterations specified by the user are complete

PREPARATION OF THE CAD MODEL

The dimensions of the computational domain which consist of Helical coil tube heat exchanger. After this process, the constraints are applied and this way the model is achieved in ANSYS 15.0. Table 1 & Table 2 shows the parameters of helical coil heat exchanger tube.

Table 1: Geometric Parameters of Helical coil heat exchanger tube

Inner Diameter (m)	Outer Diameter (m)	Coil Diameter (m)	Pitch (m)
63	91	86	0.115

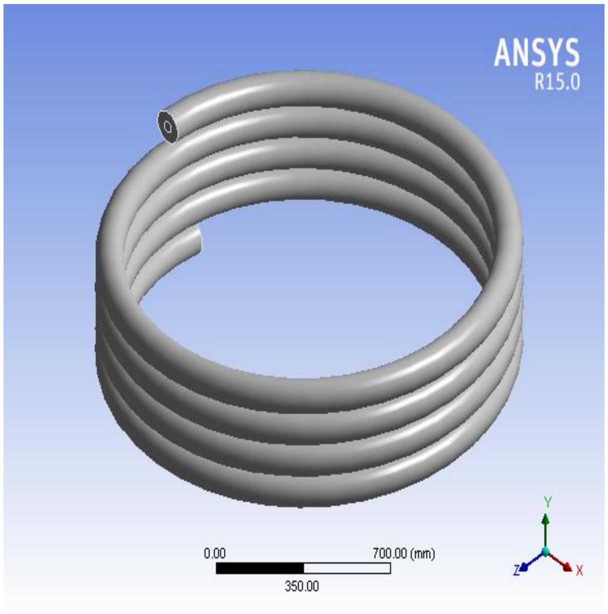

Figure 1: 3D Model of Helical Coil heat exchanger.

MESHING OF THE DOMAIN

After modelling the geometry in ANSYS, it was generated to ANSYS workbench design modular for its discretization. Meshing is dividing the complete geometry of interest into small parts. Mesh density varies based upon the assigned refinement factor. Mesh is the key part of a high quality convergence. There are 3 types of meshing. These are Hexahedral Cartesian, Hexahedral Unstructured and Tetrahedral meshes. Hexahedral Cartesian mesher generates totally structured meshes. It’s appropriate for free kind of geometries. But inappropriate for models where curved surfaces exist. Hexahedral Unstructured mesher creates nodes and parts of hexahedral cells dominantly and tetrahedral cells wherever necessary. Tetrahedral mesmerize is designed for terribly difficult geometries where the other two can't be used. For models involving spheres or ellipsoids hexahedral mash are useless.

For present case, hexahedral unstructured mesh has been used. Total number of nodes is 27271 and elements is 5805 which have been employed for the analysis of modified Helical coil heat exchanger tube. The results of the grid refinement study show that the considered mesh for the Helical coil tube heat exchanger would provide satisfactory numerical accuracy.

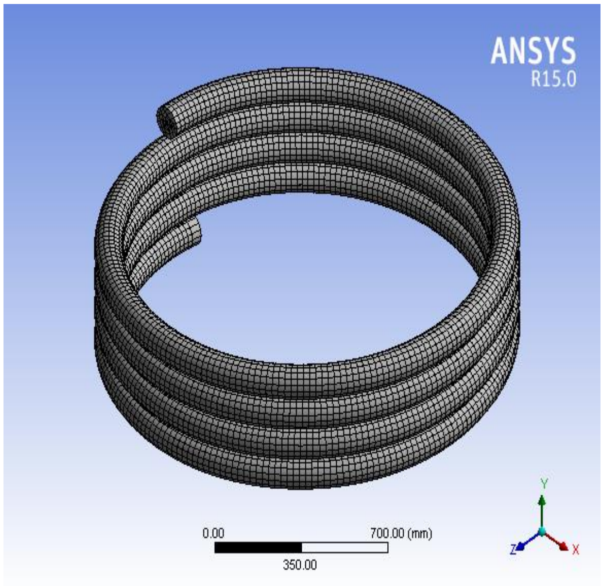

Figure 2: Meshing of Helical coil heat exchanger tube

4. EXPERIMENTAL RESULT

A three-dimensional model has been developed to investigate heat transfer in the helical coil tube heat exchanger for determining heat transfer rate. A series of numerical calculations have been conducted using commercial CFD code FLUENT15.The results are presented in order to show the effects of overall heat transfer coefficient with respect to mass flow rate in the helical coil heat exchanger tube. The Numerical result has been compared with the base paper results[6].

OPTIMIZATION OF HELICAL COIL TUBE HEAT EXCHANGER WITH CUO NANOFLUID

Table 1 shows the relative humidity values w.r.t mass flow rate inlet and mass flow rate outlet from table, it is determined that relative humidity within the Helical coil heat exchanger tube decreases w.r.t. mass flow rate inlet and mass flow rate outlet.

Table 1 Shows the overall heat transfer coefficient comparison with water and CuO.

Comparison Results of Overall Heat Transfer	
Water and CuO	
Overall Heat Transfer W/m2K (CuO)	Overall Heat Transfer W/m2K (Water)
32.6	31.8
44.8	42.9
52.6	48.6
77.9	75.8
95.8	93.2
117.9	110.8
142.8	135
158.7	155.9

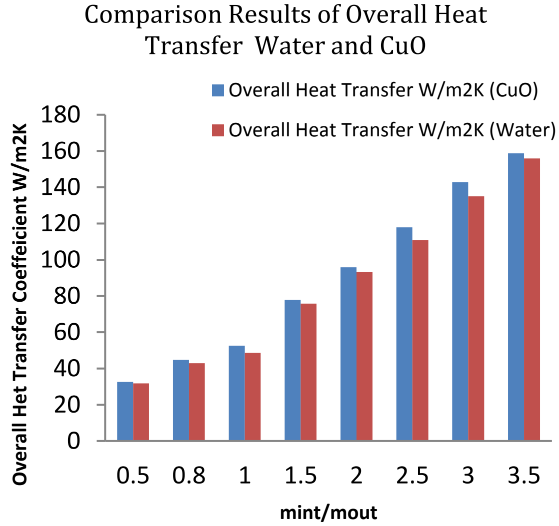

Figure 3: shows the overall heat transfer coefficient comparison with water and CuO.

Overall heat transfer coefficient of Helical coil heat exchanger tube under different mass flow rate between CuO and water varies from 5% to 15%. If the density of water temperature distribution decreases.

NUSSELT NO. VARIATION ON HELICAL COIL TUBE HEAT EXCHANGER

Table 2 shows the simulation results of variation in nusselt no. for the Helical coil heat exchanger tube with respect to mass flow rate inlet and mass flow rate outlet.

Comparison Results of Nusselt no.	
Water and CuO	
Nusselt no. (CuO)	Nusselt no. (Water)
2.3	1.8
3.4	2.9
4.2	3.6
5.8	5.1
7	6.3
7.9	7.2
9.3	8.4
10.4	9.6

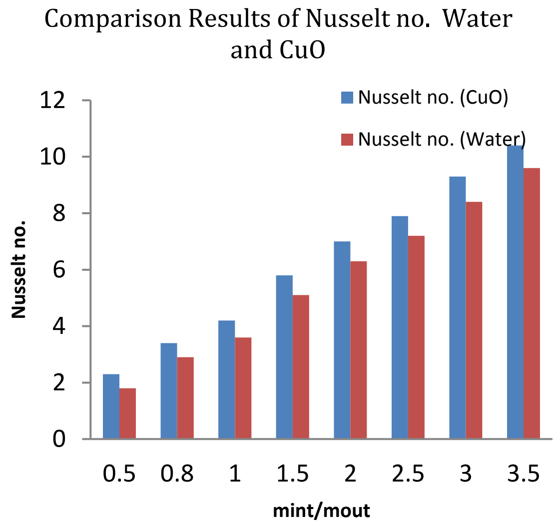

Figure 4: Simulation Result of variation in nusselt no. for the helical coil tube heat exchanger with variation in mass flow rate inlet and mass flow rate outlet.

Figure 4 shows the simulation result of helical coil heat exchanger tube to determine nusselt no., Numerical results are slightly above than Base paper result values, the deviation almost constant.

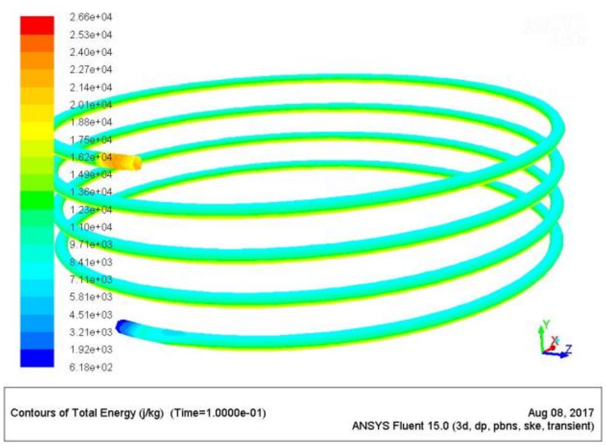

Figure 5: Total energy variations on helical coil heat exchanger tube on CuO nanofluid.

OVERALL COMPARISON OF HEAT TRANSFER COEFFICIENT AND NUSSELT NO.

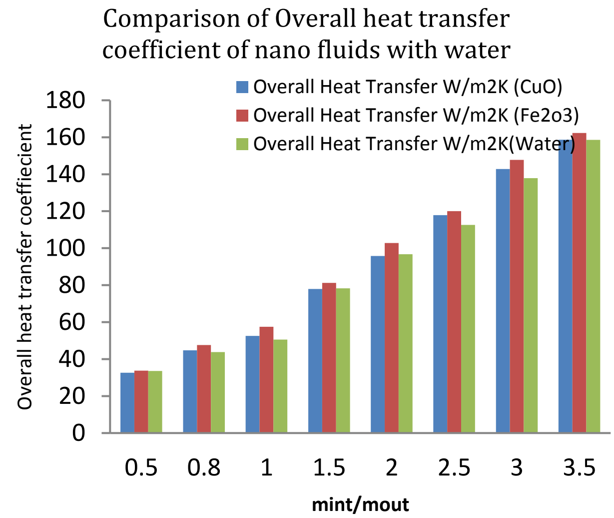

Figure 6: Shows the Variation of overall transfer coefficient. w.r.t mass flow rate inlet and mass flow rate outlet of different nano fluids and water.

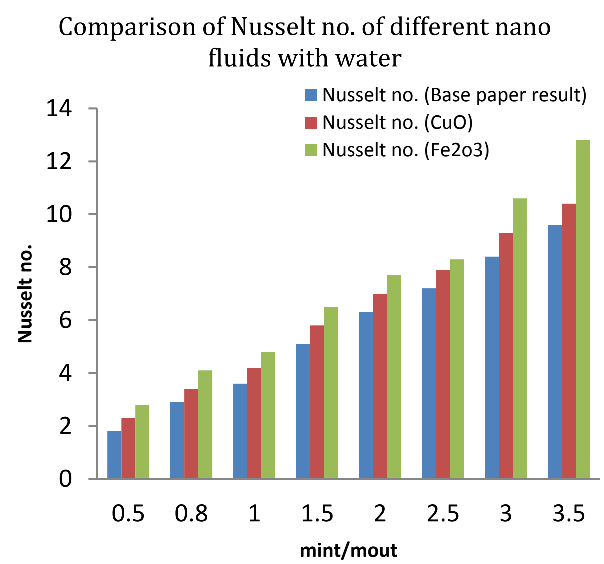

Figure 7: Shows the Variation of nusselt no. w.r.t mass flow rate inlet and mass flow rate outlet of different nano fluids and water.

1. Heat transfer within a fluid occurs due to density difference within it.
2. Nusselt no. average value of temperature obtained Numerically varies by about 6.8% to the experimental results. For forced convection average value of temperature is large about 13% as compared to base paper results.

3. Variation in overall comparison of heat transfer coefficient and nusselt no. Varies by 0.78% and 0.73% respectively.

5. CONCLUSION AND FUTURE WORK

Computational model has been developed in ANSYS 15.0 and analysis has been done in Fluent 15.0. Numerical results are in good agreement with Base paper result results. The internal consistency of the results confirms the validity of the CFD model. From results, higher value of temperature, overall heat transfer coefficient, and nusselt is found out for ferrous oxide nano fluid than water and CuO. Fe2O3 shows maximum heat transfer rate than water and CuO thus ferrous oxide nano fluid shows more convergence than other mathematical models.

CFD analysis can also be done for other heat exchangers using same Methodology and Boundary condition. Nano fluid for heat exchanger can be further varied and its effect can be studied on its performance. Same methodology can be used to check the performance of series, cross flow and parallel flow heat exchanger also.

REFERENCES

[1] Gabriela Huminic and Angel Huminic "Heat transfer and entropy generation analyses of nanofluids in helically coiled tube-in-tube heat exchangers", International Communications in Heat and Mass Transfer, 2016, Pp 118–125.

[2] M.A. Khairul, R. Saidur, M.M. Rahman, M.A. Alim, A. Hossain and Z. Abdin "Heat transfer and thermodynamic analyses of a helically coiled heat exchanger using different types of nanofluids", International Journal of Heat and Mass Transfer, 2013, Pp 398–403.

[3] Mohammad Mehrali, Emad Sadeghinezhad, MarcA. Rosenc, Amir Reza Akhiania, Sara Tahan Latibari, Mehdi Mehrali and Hendrik Simon Cornelis Metselaar "Heat transfer and entropy generation for laminar forced convection flowof graphene nanoplatelets nanofluids in a horizontal tube", International Communications in Heat and Mass Transfer, 2015, Pp 23–31.

[4] M.A. Khairul, M.A.Alima, I.M. Mahbubul, R. Saidura, A. Hepbasli and A.Hossain "Heat transfer performance and exergy analyses of a corrugated plate heatexchanger using metal oxide nanofluids", International Communications in Heat and Mass Transfer, 2014, Pp 8–14.

[5] M.R. Sohel, R. Saidur, N.H. Hassan, M.M. Elias, S.S. Khaleduzzaman and I.M. Mahbubul "Analysis of entropy generation using nanofluid flow through the circular micro channel and minichannel heat sink", International Communications in Heat and Mass Transfer, 2013, Pp 85–91.

[6] Abhinav Gupta, Ravi Kumar and Akhilesh Gupta "Condensation of R-134a inside a helically coiled tube-in-shell heat exchanger", Experimental Thermal and Fluid Science, 2014, Pp 279–289.

[7] M. Farzaneh-Gord, H. Ameri and A. Arabkoohsar "Tube-in-tube helical heat exchangers performance optimization by entropy generation minimization approach", Applied Thermal Engineering, 2016, Pp 1279–1287.

[8] Hamed Sadighi Dizaji, Samad Jafarmadar and Mehran Hashemian "The effect of flow, thermodynamic and geometrical characteristics on exergy loss in shell and coiled tube heat exchangers", Elsevier, 2015, Pp 678-384.

[9] R. Ellahi, M. Hassan and A. Zeeshan "Shape effects of nanosize particles in Cu—H_2O nanofluid on entropy generation", International Journal of Heat and Mass Transfer, 2015, Pp 449–456.

[10] Lorenzo Santini, Andrea Cioncolini, Matthew T. Butel and Marco E. Ricotti "Flow boiling heat transfer in a helically coiled steam generator for nuclear power applications", International Journal of Heat and Mass Transfer, 2016, Pp 91–99.

[11] Bharathwaj V, Markan A, Atrey M, Neumann H and Ramalingam R "Fiber Bragg Gratings for distributed cryogenic temperature measurement in a tube in tube helically coiled heat exchanger", IEEE, 2014, Pp 1-4.

[12] Zan Wu, Lei Wang, Bengt Sundén and Lars Wadsö "Aqueous carbon nanotube nanofluids and their thermal performance in a helical heat exchanger", Applied Thermal Engineering, 2016, Pp 364–371.

[13] Jie Yang and Wei Liu "Numerical investigation on a novel shell-and-tube heat exchanger with plate baffles and experimental validation", Energy Conversion and Management, 2015, Pp 689–696.

[14] Changzhao Pan, Yuan Zhou and Junjie Wang "CFD study of heat transfer for oscillating flow in helically coiled tube heat-exchanger", Computers and Chemical Engineering, 2014, Pp 59–65.

[15] Hamed Sadighi Dizaji, Shahram Khalilarya, Samad Jafarmadar, Mehran Hashemian and Mohammad Khezri "A comprehensive second law analysis for tube-in-tube helically coiled heat exchangers", Experimental Thermal and Fluid Science, 2016, Pp 118–125.

[16] Andrew Michael Fsadni and Justin P.M. Whitty "A review on the two-phase heat transfer characteristics in helically coiled tube heat exchangers", International Journal of Heat and Mass Transfer, 2016, Pp 551–565.