e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 13
October 2017

Cfd Analysis of Helically Coiled Tube for Compact Heat Exchangers

NAJEER MATTIPATI[1], K.CHANDRA SEKHAR[2]

[1]M-Tech (THERMAL ENGG), Department of Mechanical Engineering, QIS College of Engineering and Technology, Ongole.

[2] Associate Professor, M.TECH(Ph.D), Department of Mechanical Engineering, QIS College of Engineering and Technology, Ongole

ABSTRACT

A helically coil-tube heat exchanger is generally applied in industrial applications due to its compact structure, larger heat transfer area and higher heat transfer capability, etc. The importance of compact heat exchangers (CHEs) has been recognized in aerospace, automobile, gas turbine power plants, and other industries for the last 60 years or more due to several factors as mentioned above. However flow and heat transfer phenomena related to helically coil-tube heat exchanger are very sophisticated.

A computational fluid dynamics (CFD) methodology using ANSYS FLUENT 13.0 is used here to investigate effects of different curvature ratio on the heat transfer characteristics in a helically coil-tube. Simulation has been done for different curvature ratios of a helical coil tube by varying different inlet conditions like velocity Based on the simulation results, the complicated phenomena occurred within a helical coil-tube can be reasonably captured, including heat transfer behaviors from the entrance region, etc.

For all the cases considered in this work, heat transfer coefficient, Nusselt number, pressure drop, Colburn factor and fRe are being computed and studied to analyze the heat transfer characteristics of a helical coil tube.

INTRODUCTION

Heat exchanger is a device that continuously transfers heat from one medium to other medium in order to carry process energy.

Heat exchangers are used in various systems for:

a) Recovering heat directly from one flowing medium to another or via a storage system, or indirectly via a heat pump or heat transformer.
b) Heating or cooling a process steam to the required temperature for a chemical reaction (this can also be direct or indirect).
c) Enabling, as an intrinsic element, a power, refrigeration or heat pumping process, that is interchanging heat between a hot source or steam with the working fluid and with the low temperature heat sink (or source).

For efficiency, heat exchangers are designed to maximize the surface area of the wall between the two fluids, while minimizing resistance to fluid flow through the exchanger. The exchanger's performance can also be affected by the addition of fins or corrugations in one or both directions, which increase surface area and may channel fluid flow or induce turbulence.

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 13
October 2017

The invention of high speed digital computers, combined with the development of accurate numerical methods for solving physical problems, has revolutionized the way we study and practice fluid dynamics and heat transfer. This approach is called Computational Fluid Dynamics or CFD in short, and it has made it possible to analyze complex flow geometries with the same ease as that faced while solving idealized problems using conventional methods. CFD may thus be regarded as a zone of study combining fluid dynamics and numerical analysis. Historically, the earlier development of CFD in the 1960s and 1970s was driven by the need of the aerospace industries. Modern CFD, however, has applications across all disciplines – civil, mechanical, electrical, electronics, chemical, aerospace, ocean, and biomedical engineering being a few of them. CFD substitutes testing and experimentation, and reduces the total time of testing and designing.

PROBLEM FORMULATION

A helical pipe with 4 turns is taken as the model for the analysis as shown in Fig. 3.1. The coil diameter (D) is taken as 300 mm and total length of the pipe (L) is 3.77 m. The pipe diameter (d) of the model shown in Fig. 3.1 is 10 mm. But, in the analysis four different values of pipe diameter are taken, keeping coil diameter as well as length constant, to see the effect of change in curvature ratio (d/D) on the heat transfer and fluid flow characteristics of a helical pipe. The fluid properties are assumed to be constant.

Fig 3.1 Model of helical pipe

After creating four different geometric models, each model was analyzed for boundary conditions of constant wall temperature and constant wall heat flux and that too for both type of fluid flow conditions i.e. laminar fluid flow and turbulent fluid flow and then results were compared for each case.

Governing Equations

Applying boundary conditions, the governing equations for convective heat transfer are as follows:

Continuity equation $$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$$

Navier-Stokes field equations (Only x-direction equation is given below)

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=\rho X-\frac{\partial p}{\partial x}+\frac{1}{3}\mu\frac{\partial}{\partial x}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)+\mu\nabla^2 u$$

Energy equation

$$\rho C_p\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=\left(u\frac{\partial p}{\partial x}+v\frac{\partial p}{\partial y}+w\frac{\partial p}{\partial z}\right)+k\nabla^2 T+\mu\emptyset$$

where is the Rayleigh dissipation function and is given by

$$\emptyset=2\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial w}{\partial z}\right)^2\right]+\left[\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2+\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)^2+\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)^2\right]$$
$$-\frac{2}{3}\left[\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right]^2$$

Heat transfer coefficient $h=\frac{-k\frac{\partial T}{\partial x}}{T_w-T_f}$

Nusselt number

$$Nu=\frac{-\frac{\partial T}{\partial x}d_h}{T_w-T_f}$$

Critical Reynolds number as per the correlation given by Schmidt (1967) $Re_{cr}=2300\left[1+8.6\left(\frac{d}{D}\right)^{0.45}\right]$

Friction factor $f=\frac{2\Delta p d}{\rho L V^2}$

Colburn factor $j=\frac{Nu_d}{Re_d Pr^{1/3}}$

Boundary Conditions

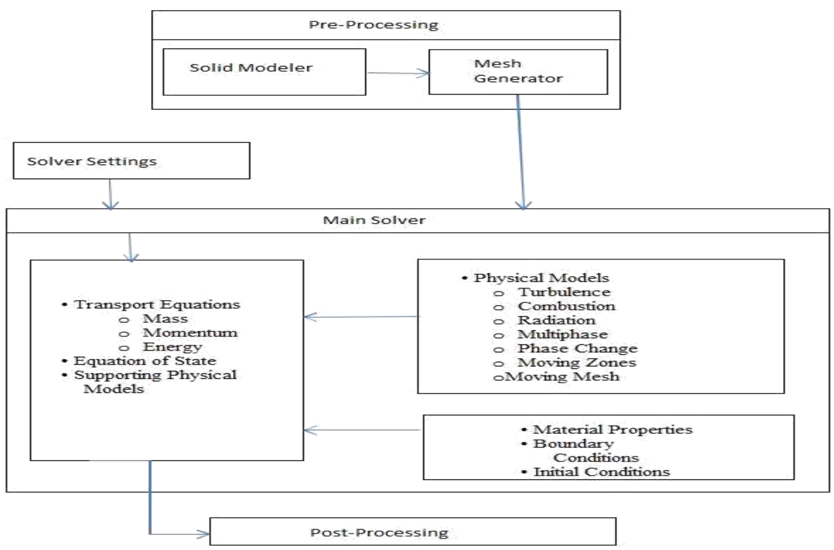
The analysis of the model has been done under two sections.

i) Effect of curvature ratio with variable velocity i.e. mass flow rate: The velocities of working fluid assumed at the inlet are 0.6m/s, 0.8m/s, 1m/s, 1.2m/s respectively.
ii) Effect of curvature ratio with variable inlet pressure: Four different gauge pressures are assumed at the inlet. They are 5000 N/m^2, 10000 N/m^2, 15000 N/m^2 and 20000 N/m^2.

In the work reported here, water-liquid is taken as the working fluid for the analysis. Fluid properties are assumed to be constant with temperature. The properties of water-liquid considered for the analysis is given in table 3.1

Further analysis has been done for two different wall boundary conditions. In the constant wall heat flux boundary conditions, for both the sections and all the four geometric models, wall heat flux is assumed to be 20000 W/m^2 and in the constant wall temperature boundary condition, wall temperature of the helical pipe is assumed to be 300 K. The inlet temperature of the fluid is taken as 360 K and pressure at the outlet to be 1atm.

Properties of water



CFD Programs

The development of affordable high performance computing hardware and the availability of user-friendly interfaces have led to the development of commercial CFD packages. Before these CFD packages came into the ordinary use, one had to write his own code to carry out a CFD analysis. The

Description	Symbol	Value	Units
Density	ρ	1000	kg/m^3
Dynamic Viscosity	μ	0.001003	kg/ms
Specific Heat	C_p	4182	J/kgK
Thermal Conductivity	k	0.6	W/mK

programs were usually different for different problems, although some part of the code of one program could be used in another. The programs were inadequately tested and reliability of the results was often questioned. Today, well tested commercial CFD packages not only have made CFD analysis a routine design tool in industry, but are also helping the research engineer in focusing on the physical system more effectively.

All established CFD software contain three elements (i) a pre-processor, (ii) the main solver, and (iii) a post-processor

The Pre-Processor

Pre-processing is the first step of CFD analysis in which the user defines the modelling objectives,

(a) identifies the computational domain, and
(b) designs and creates the grid system

The process of CFD modelling starts with an understanding of the actual problem and identification of the computational domain. This is followed by generations of the mesh structure, which is the most important portion of the pre-processing activity. It is believed that more than 50% of the time spent by a CFD analyst goes towards mesh generation. Both computation time and accuracy of solution depend on the mesh structure. Optimal grids are generally non-uniform – finer in areas where large variation of variables is expected and coarser in regions where relatively little changes is expected. In order to reduce the difficulties of engineers and maximize productivity, all the major CFD programs include provision for importing shape and geometry information from CAD packages like AutoCAD and I-DEAS, and mesh information from other packages like GAMBIT.

The Main Solver

The solver is the heart of CFD software. It sets up the equations which are selected according to the options chosen by the analyst and grid points generated by the pre-processor, and solves them to compute the flow field. The processes incorporate the following tasks:

- Selecting appropriate physical model,
- Defining material properties,
- Prescribing boundary conditions,
- Providing initial solution,
- Setting up solver controls,

Solving equation set, and Saving results.

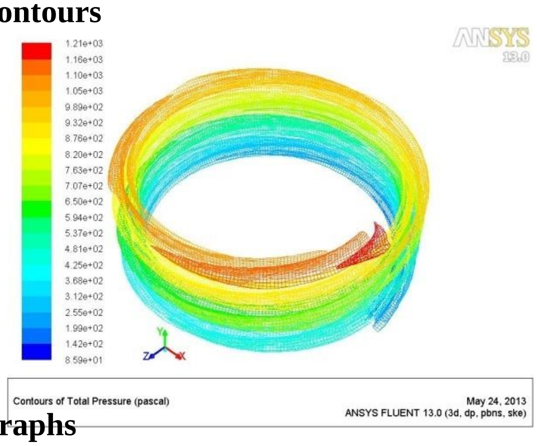
Once the model is completely set, the solution is initialized consequently calculation starts and intermediate results can be monitored at every time step from iteration to iteration. The progress of the solution process get displayed on the screen in terms of the residuals, a measure of the extent to which the governing equations are not satisfied.

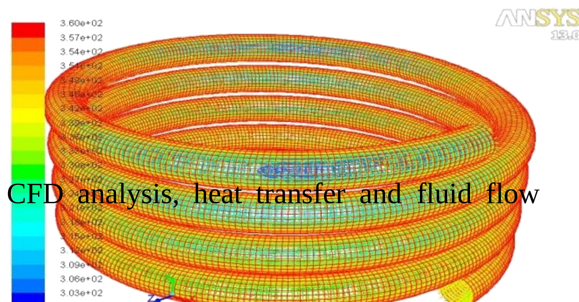
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 13
October 2017

Results and Discussion

The heat transfer and flow characteristics of a helical pipe can be visualized from the contour diagrams of pressure and temperature distribution, values of Nusselt number and friction factor which have been tabulated, and the graphs of heat transfer coefficient, Nusselt number, pressure difference and fRe for various heat transfer and flow conditions which have been plotted using ANSYS FLUENT 13.0.

Contours





Graphs

From the plotted graphs using values obtained from the CFD analysis, heat transfer and fluid flow characteristics can be easily visualized.

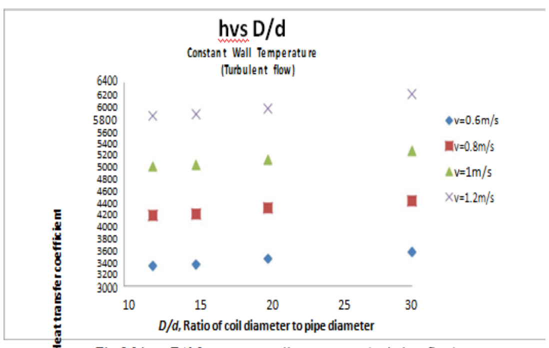
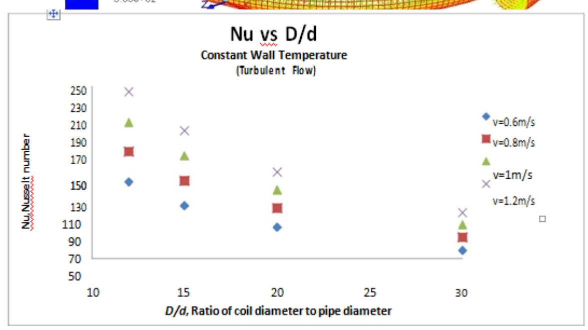
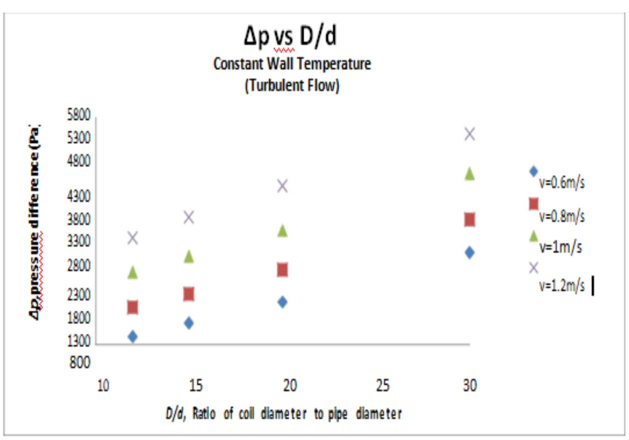


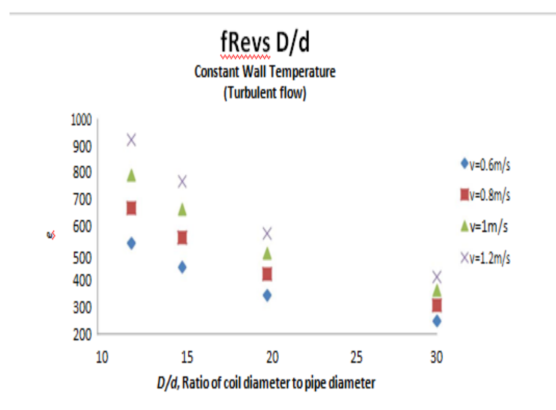
Fig.5.3 h vs. D/d for constant wall temperature (turbulent flow)

h vs. D/d for constant wall temperature (turbulent flow)



Nu vs. D/d for constant wall temperature (turbulent flow)





Δp vs. D/d for constant wall temperature (turbulent flow)   fRe vs. D/d for constant wall temperature (turbulent flow)

shows heat transfer and fluid flow characteristics of helical pipe for constant wall temperature boundary condition and turbulent flow. As can be seen from fig.5.4 that as the curvature ratio (ratio of pipe diameter to coil diameter) increases i.e. D/d ratio decreases, Nusselt number increases which means a higher curvature ratio will give better heat transfer

CONCLUSIONS AND FUTURE SCOPE

Through the CFD methodology, this work investigates the flow and heat transfer phenomena in a helical pipe. Effects of inlet mass flow rate, inlet pressure and curvature ratio on these characteristics have been also studied. Several important conclusions could be drawn from the present simulations and would be presented as follows:

- It is visible from the results that Nusselt Number depends on curvature ratio. It is increasing with increase in curvature ratio. In addition, the value of Nu no. was found to increase with increase in mass flow rate (i.e. inlet velocity), which can also be confirmed by experiments.
- It can also be visualized from the results that friction factor is more in turbulent flow compared to laminar flow and also results shows their dependency on curvature ratio under variable Reynolds number.
- Nusselt number as well as friction factor is increasing with increase in curvature ratio. So, there must be an optimum value for which helical pipe will give best performance.
- For laminar flow, Nusselt number almost remains constant with slight increase in inlet velocity as well as with increase in inlet pressure.
- It seems from the results that higher curvature ratio of helical pipe will have better heat transfer rate.
- As predicted helical pipe has better heat transfer performance as compared to a straight pipe.

Future Scope

The works which are required to be done in future are:

- To numerically model a helically coil tube heat exchanger using CFD analysis and optimize the curvature ratio using Dean number and Colburn factor for boundary conditions of constant wall heat flux and constant wall temperature for both laminar flow and turbulent flow.
- To design an optimized and more efficient helical coil tube heat exchanger.

performance. It can also be seen from fig.5.5 that with increase in curvature ratio, pressure loss is also decreasing, so we can say that a higher curvature ratio is better for good performance of helical pipe.

References

[1] Abdulla, M.A. A four region, moving boundary model of a once through, helical oil team generator. Ann Nucl Energy, 21(1994): 541–562.
[2] Akiyama, M. and Cheng, K.C. Boundary vorticity method for laminar forced convection heat transfer in curved pipes. Int J Heat Mass Transf, 14 (1971): 1659–1675.
[3] Bai, B., Guo, L., Feng, Z. and Chen, X. Turbulent heat transfer in a horizontally coiled tube. Heat Transf Asian Res, 28 (1999): 395–403.
[4] Berger, S.A., Talbot, L. and Yao, L.S. Flow in curved pipes. Ann Rev Fluid Mech, 15 (1983): 461–512.
[5] Darvid, A.N., Smith, K.A., Merril, E.W. and Brain, P.L.T. Effect of secondary fluid motion on laminar flow heat transfer in helically coiled tubes. AICHE J, 17 (1971): 1142–1222.
[6] Dittus, F.W. and Boelter, L.M.K. Publication on Engineering (University of California Press, Berkeley, CA), (1930). p. 443
[7] Flavio, C.C.G., Raquel, Y.M., Jorge, A.W.G. and Carmen, C.T. Experimental and numerical heat transfer in a plate heat exchanger. ChemEngSci, 61 (2006): 7133-7138
[8] Futagami, K. and Aoyama, Y. Laminar heat transfer in helically coiled tubes. Int J Heat Mass Transf, 31 (1988): 387–396