CFD ANALYSIS OF HELICALLY COILED TUBE FOR COMPACT HEAT EXCHANGERS

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF

Master of Technology
in
Mechanical Engineering
with Specialization in
"Thermal Engineering"

By

Pooja Jhunjhunwala

Department of Mechanical Engineering
National Institute of Technology
Rourkela

June 2013

CFD ANALYSIS OF HELICALLY COILED TUBE FOR COMPACT HEAT EXCHANGERS

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF

Master of Technology
in
Mechanical Engineering
with Specialization in
"Thermal Engineering"

By
Pooja Jhunjhunwala

Under the guidance of
Prof.A.K.Satapathy

Department of Mechanical Engineering
National Institute of Technology
Rourkela
June 2013

National Institute of Technology
Rourkela

CERTIFICATE

This is to certify that the thesis entitled, **"CFD ANALYSIS OF HELICALLY COILED TUBE FOR COMPACT HEAT EXCHANGER"** submitted by **Miss Pooja Jhunjhunwala** in partial fulfillment of the requirements for the award of Master of Technology Degree in **Mechanical Engineering** with specialization in **Thermal Engineering** at the National Institute of Technology, Rourkela (Deemed University) is an authentic work carried out by her under my supervision and guidance.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other University/ Institute for the award of any degree or diploma.

Date:

Prof. ASHOK KUMAR SATAPATHY
Department of Mechanical Engineering
National Institute of Technology
Rourkela – 769008

ACKNOWLEDGEMENT

I would like to express my deep sense of gratitude and respect to my supervisor Prof. Ashok Kumar Satapathy for his excellent guidance, suggestions and constructive criticism. Working under his supervision greatly contributed in improving quality of my research work and in developing my engineering and management skills. I would also like to thank NIT Rourkela and MHRD of India for providing me research facilities and opportunity.

I am extremely fortunate to be involved in such an exciting and challenging research project. It gave me an opportunity to work in a new environment of Fluent. This project has increased my thinking and understanding capability.

I would like to express my thanks to all my friends, all staffs and faculty members of mechanical engineering department for making my stay in N.I.T. Rourkela a pleasant and memorable experience.

I would like to thank all whose direct and indirect support helped me in completing my thesis in time.

Lastly I would like to convey my heartiest gratitude to my parents and my sisters for their unconditional love and support.

Pooja Jhunjhunwala

Roll No. 211ME3189

Department of Mechanical Engg

National Institute of technology

CONTENTS

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-inlet and pressure-inlet for different flow and heat transfer conditions. 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.

List of Tables

List of Figures

ABBREVIATIONS & ACRONYMS

A_c flow area, m^2

A_s surface area, m^2

C_p specific heat, J/kgK

d diameter of pipe, mm

d_h hydraulic diameter, mm

D coil diameter, mm

De Dean Number

f friction factor

G mass velocity, m/s

h heat transfer coefficient, W/m^2K

H pitch of coil, mm

j Colburn factor

k thermal conductivity, W/mK

L length of pipe, m

m mass flow rate, kg/s

n number of turns

Nu Nusselt Number

p pressure, N/m^2

Pr Prandtl Number

Q heat flux, W/m^2

Re Reynolds Number

St Stanton Number

T Temperature, K

u, v, w velocity along X-axis, Y-axis and Z-axis, m/s

V flow velocity, m/s

V_s wetted volume, m^3

x, y, z coordinates

X body force

Greek symbols

β surface area density, m^2/m^3

ρ density, kg/m^3

μ dynamic viscosity, kg/ms

φ Rayleigh dissipation factor

Subscripts

cr critical

d diameter

f fluid

w wall

CHAPTER-1

INTRODUCTION

1.1 Heat Exchanger

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.

1.2 Compact Heat Exchanger

A compact heat exchanger can be defined as heat exchanger which has area density greater than $700m^2/m^3$ for gas or greater than $300m^2/m^3$ when operating in liquid or two-phase streams.

The concept behind compact heat exchanger is to decrease size and increase heat load which is the typical feature of modern heat exchanger. The importance of compact heat exchangers

(CHEs) has been appreciated in aerospace, automobile, cryogenics, gas turbine power plant, and other industries for the last 60 years or more. This is due to various factors, for example packaging constraints, sometimes high performance requirements, low cost, and the use of air or gas as one of the fluids in the exchanger.

The other driving factor from last three decades for heat exchanger design has been reducing energy consumption for operation of heat exchangers and minimizing the capital investment in industries. Consequently, in process industries where not-so-compact heat exchangers were mostly common, the use of helical coil-tube heat exchangers and other CHEs has been increasing owing to some of the inherent advantages mentioned above. In addition, CHEs offer the reduction of floor space, decrease in fluid inventory in closed system exchangers, and tighter process control with liquid and phase change working fluids.

1.3 Basic Aspects of Compactness

There are basically two types of aspects of compactness. They are:

a) **Geometrical aspects:-**

The basic parameter describing compactness is the hydraulic diameter d_h , defined as

$$d_h = \frac{4A_cL}{A_s}$$

where, A_c = flow area

and A_s = surface area

For some types of geometries, the flow area varies with flow length, so for these there is an alternative definition

$$d_h = \frac{4V_s}{A_s}$$

where, V_s = enclosed (wetted) volume

This second definition helps us to link hydraulic diameter to the surface area density β, which is A_s/V , also called as a measure of compactness.

A commonly accepted lower threshold value for β is $300 m^2/m^3$, which for a typical porosity of 0.75 gives a hydraulic diameter of about 10 mm.

b) **Heat Transfer Aspects of Compactness:-**

The heat transfer coefficient h is generally expressed in compact surface terminology, in terms of the dimensionless j, or Colburn factor by the definition

$$j = \frac{Nu}{RePr^{1/3}} = StPr^{2/3}$$

where, Nu (Nusselt number) = $\frac{hd_h}{k}$, and

St (Stanton number) = $\frac{h}{GC_P}$

where G = mass velocity

For a single side a specified heat load $\dot{Q}$, is given by heat transfer rate equation

$$\dot{Q} = hA_s\Delta T = \dot{m}C_P(T_2 - T_1) ,$$

neglecting for comfort the influences of wall resistance and surface efficiency on h.

Therefore, $\dot{Q} = \frac{4hV_S\Delta T}{d_h}$ since $(A_s = \frac{4V_S}{d_h})$

Thus for a specified heat load $\dot{Q}$, to reduce the volume we have to increase the ratio h/d_h. The choice therefore is to increase heat transfer coefficient h or to decrease hydraulic diameter i.e. to increase compactness, or both.

1.4 Helical Coil-Tube Heat Exchanger

Recent developments in design of heat exchangers to fulfill the demand of industries has led to the evolution of helical coil heat exchanger as helical coil has many advantages over a straight tube. So, it has become necessary to study and analyze helical coil in a broader sense.

1.4.1 Advantages:

- Heat transfer rate in helical coil are higher as compared to a straight tube
- Compact structure
- Larger heat transfer area

1.4.2 Applications:

- Heat exchangers with helical coils are widely used in industrial applications such as power generation, nuclear industry, process plants, refrigeration, heat recovery systems, food industry, etc.

- Helical coil heat exchanger is used for residual heat removal system in islanded or barge mounted nuclear reactor system, where nuclear energy is used for desalination of seawater
- In cryogenic applications including LNG plant

1.5 Objectives of Work

The objective of the present work is to study the heat transfer characteristics of a helical coil with the variation in curvature ratio (d/D) or Dean Number (De). This analysis has to be done for boundary conditions of both constant wall heat flux and constant wall temperature and also for different flow conditions i.e. laminar flow and turbulent flow. After that comparison of the performance of a helical coil with that of a straight tube has to be done.

1.6 Organization of the Thesis

This thesis comprises of six chapters excluding references.

Chapter 1 gives the brief introduction of heat exchanger, compact heat exchanger, aspects of compactness and helical coil heat exchanger and with the objective of work.

In chapter 2, I have given a brief literature review about the topic and research which are related to my present work.

Chapter 3 deals with the introduction of my problem with its governing equations and boundary conditions.

In chapter 4 CFD modeling of the problem has been done.

Chapter 5 deals with the results and discussions of my research work for all the considerer boundary conditions.

Chapter 6 gives the conclusion and scope of future work.

CHAPTER-2

LITERATURE REVIEW

2.1 Literature Survey:

In a wide range of literature it has been reported that heat transfer rates in helical coils are higher as compared to a straight tube because of the secondary flow pattern in planes normal to the main flow. This secondary flow occurs because of the difference in velocity, and its pattern changes with the Dean number of the flow. The fluid streams in the outer side of the pipe moves faster than the fluid streams in the inner side of the pipe due to the effect of curvature which results in difference in velocity. Many researchers have reported that a complex flow pattern exists inside a helical pipe which leads to the enhancement in heat transfer. The centrifugal force results in the development of secondary flow (Dravid et al.,1971) and this centrifugal force is governed by the curvature of the coil while the torsion to which the fluid is subjected to is affected by the pitch or helix angle of coil. Dean number is used to characterize the flow in a helical pipe (Jayakumar et al.,2008). So, in my investigation I varied curvature ratio i.e. Dean number to analyze the performance of helical pipe for various boundary conditions.

A considerable amount of work has been done on the flow and heat transfer of fluid inside helically coiled tubes as reported in literature. In spite of numerical and experimental studies that have been done in relation to helical coil tube, there are not many investigations on the behavior of helical coil tube with change in curvature ratio for any boundary condition. Jayakumar et al, (2008) had done experimental and CFD estimation of heat transfer in helically coiled heat exchangers for temperature dependent properties and conjugate heat transfer. Here, in my work I have assumed that fluid properties are constant and have analyzed the heat transfer characteristics for both constant wall heat flux and constant wall temperature conditions.

Jayakumar and Grover (1997) have investigated the performance of the residual heat removal system, which uses a helically coiled heat exchanger, for various process parameters. Jayakumar et al., (2002) had further extended that work to find out the stability of operation of such a system when the barge on which it is mounted is moving.

Berger et al., (1983) have reviewed heat transfer and flow through a curved tube comprehensively first time and followed by Shah and Joshi (1987). Naphan and Wongwises (2006) had done review of flow and heat transfer characteristics in curved pipes. Many researchers have reported the heat transfer and flow characteristics of a helical pipe. But Prabhanjan et al. (2004), Berger et al. (1983), Janseen and Hoogendoorn (1978) and Ruthven (1971) have reported the heat transfer enhancement in helical coil systems. An experimental investigation on condensing heat transfer and pressure drop of refrigerant R 134a in helicoidally i.e. helical double pipe has been done by Kang et al. (2000). An experimental investigation has been done by Yamamoto et al. (1995) to study the effect of torsion on the flow in a helical tube of circular cross-section for a range of Reynolds numbers from about 500 to 20,000.

Most of the investigations on heat transfer coefficient have been done either for constant wall temperature or constant heat flux conditions (Prabhanjan et al., 2004; Shah and Joshi, 1987; Nandakumar and Masliyah, 1982) but in my research I have studied both constant heat flux and constant wall temperature conditions. The situation of constant wall temperature is idealized in heat exchangers with phase change such as condensers and the boundary condition of constant heat flux finds application in electrically heated tubes and nuclear fuel elements (Jayakumar et al., 2008). Rennie and Raghavan (2005) had conducted an experimental study of a double pipe heat exchanger. Afterward, a numerical investigation of the double pipe helical coil heat exchanger was done by Rennie and Raghavan (2006 a, b). A study for pressure drop and heat transfer in tube in tube helical heat exchanger was done by Kumar et al. (2006). However, the flow pattern is entirely different in the helically coiled tube heat exchanger than for a double pipe heat exchanger. Hence, the analysis done in my work is entirely different from those reported in earlier studies.

In this work, a numerical analysis on heat transfer characteristics of a helical coil tube with change in curvature ratio i.e. Dean number for different boundary and flow conditions has been done using ANSYS Fluent (13.0 version). Jayakumar and Grover (1997) did experimental study on helically coiled heat exchanger, but in my work analysis has been done numerically. Many previous works on flow and heat transfer related to helically coiled tubes had been done analytically. Patankar et al. (1974) have analytically investigated effects of the De number on heat transfer in helically coiled tubes for the developing and the fully-developed laminar flow. Yang et al. (1995) investigated the fully-developed laminar convective heat transfer in a helical pipe by developing a numerical model. Yang et al. (1996) further used the k-ϵ model to analyze the fully-developed turbulent convective heat transfer in a helical pipe with substantial pitch.

CFD has also been used to analyze the performance of heat exchanger. Such studies on helically coiled double pipe heat exchanger have also been carried out. Rennie and Raghavan (2005) had numerically modeled such a heat exchanger for laminar fluid flow and studied heat transfer characteristics. In the presented work, heat exchanger is modeled for both laminar fluid flow and turbulent fluid flow. In their analysis, Rennie and Raghavan (2005) have modeled the heat transfer from hot fluid to cold fluid using PHOENICS 3.3 (a CFD package) and found out the overall heat transfer coefficient for countercurrent and parallel flows. It has also been found out from the literature that the heat transfer coefficient predicted by the Dittus-Boelter equation is comparable with those calculated by Fluent, with a maximum error of 5%. Hence, we can confidently employ CFD modeling for the prediction of heat transfer coefficient.

Hence, the proposed work is different from those reported in literature and it may contribute in further improvement of the performance of helically coiled heat exchanger.

CHAPTER-3

PROBLEM FORMULATION

3.1 Introduction

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.

3.2 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 $\emptyset$ is the Rayleigh dissipation function and is given by

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

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

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

Length of the pipe

$$L = n\sqrt{H^2 + (\pi D)^2}$$

3.3 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.

Table 3.1 Properties of water

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

CHAPTER-4

CFD MODELING

4.1 Introduction

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. Fig. 4.1 gives the overview of the CFD modeling process.

4.2 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 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.

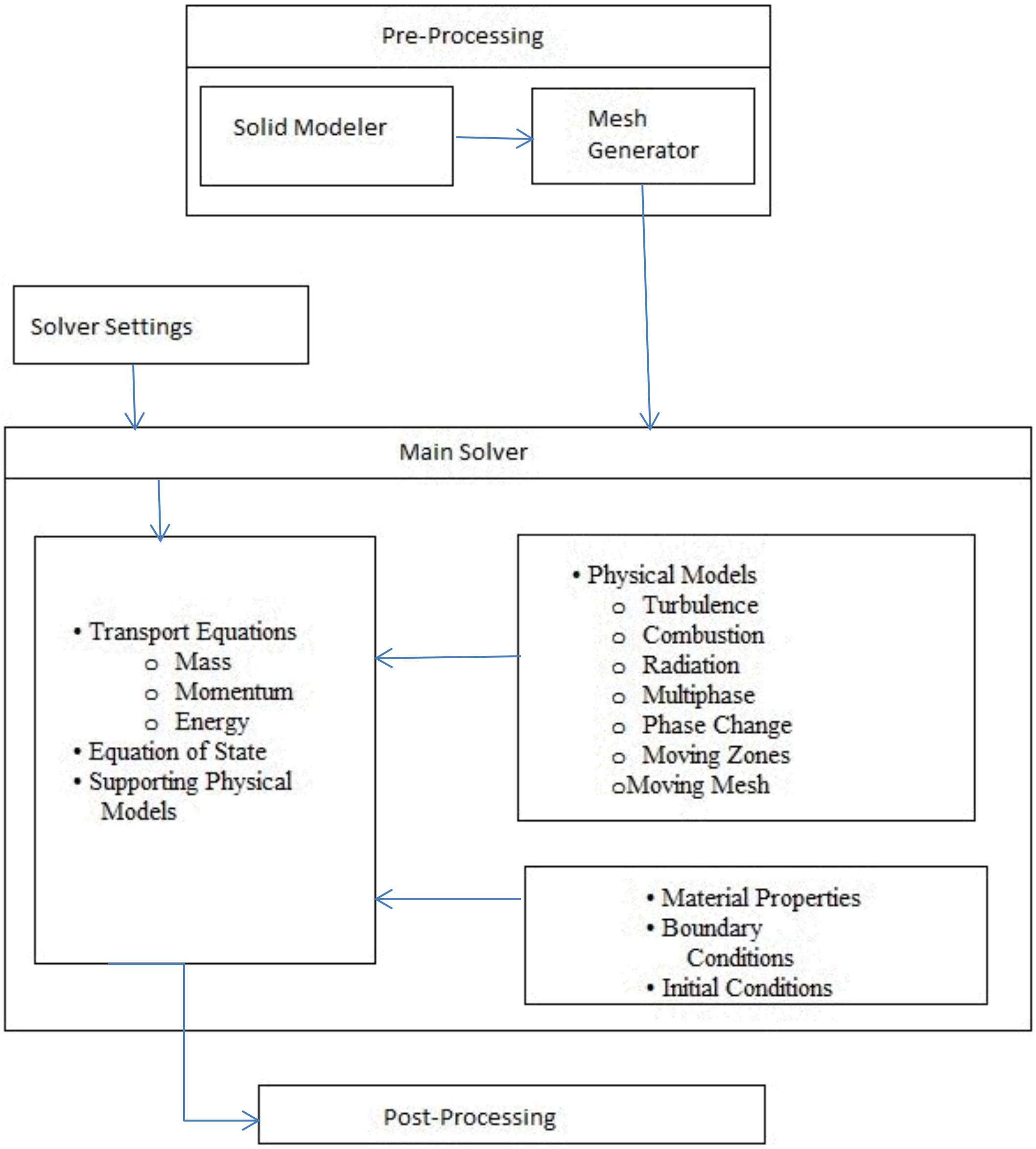

Fig.4.1 Overview of Modeling Process

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

4.2.1 The Pre-Processor

Pre-processing is the first step of CFD analysis in which the user

(a) defines the modeling objectives,

(b) identifies the computational domain, and

(c) designs and creates the grid system

The process of CFD modeling 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.

4.2.2 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 process incorporate the following tasks:

- selecting appropriate physical model,
- defining material properties,
- prescribing boundary conditions,
- providing initial solutions,

- setting up solver controls,
- setting up convergence criteria,
- 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.

4.2.3 The Post-processor

The post-processor is the last part of CFD software. It helps the user to analyze the results and get useful data. The results may be displayed as vector plots of vector quantities like velocity, contour plots of scalar variables, for example pressure and temperature, streamlines and animation in case of unsteady simulation. Global parameters like skin friction coefficient, lift coefficient, Nusselt number and Colburn factor etc. may be computed through appropriate formulas. These data from a CFD post-processor can also be exported to visualization software for better display and to software for better graph plotting.

Various general-purpose CFD packages have been published in the past decade. Important among them are: PHOENICS, FLUENT, STAR-CD, CFX, CFD-ACE, ANSWER, CFD++, FLOW-3D and COMPACT. Generally all these packages are based on the finite volume method. CFD packages have also been developed for special applications. FLOTHERM and ICEPAK for electronics cooling, CFX-TASCFLOW and FINE/TURBO for turbo machinery and ORCA for mixing process analysis are some examples. Most CFD software packages contain their own mesh generators and post processors. Some popular visualization software used with CFD packages are TECPLOT and FIELDVIEW.

4.3 Overview of FLUENT Package

FLUENT is a state-of-the-art computer program for modeling heat transfer and fluid flow in complex geometries. FLUENT provides complete mesh flexibility, solving one's flow problems with unstructured grids that can be generated about complex geometries with relative ease. Supported grid types include 2D triangular/quadrilateral. 3D FLUENT also allows user to refine or coarsen grid based on the flow solution.

FLUENT is written in the C computer language and makes full use of the flexibility and power offered by the language. As a result, true dynamic memory allocation, efficient data structures, and flexible solver control (user defined functions) are all made possible. In addition, FLUENT uses a client/server architecture, which allows it to run separate simultaneous processes on client desktop workstations and powerful computer servers, for efficient execution, interactive control, and complete flexibility of machine or operating system type.

All functions necessary to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface. The user interface is written in a language called Scheme, a dialect of LISP. The advanced user can customize and enhance the interface by writing menu macros and functions.

4.4 CFD Procedure

For numerical analysis in CFD, it requires five stages such as:

- Geometry creation
- Grid generation

- Flow specification
- Calculation and numerical solution
- Results

Based on control volume method, 3-D analysis of fluid flow and heat transfer for the helical coiled tube has been done on ANSYS FLUENT 13.0 software. All the above mentioned processes are done using the three CFD tools which are pre-processor, solver and post-processor.

4.4.1 Geometry Creation

A 3-d model of helical pipe has been created using design modeler of ANSYS as shown in fig.3.1.

4.4.2 Mesh Generation

The mesh of the model is shown in figs.4.2 and 4.3. It depicts that the domain was meshed with rectangular cells. Grid independence was studied by doing different simulation with taking different no cells.

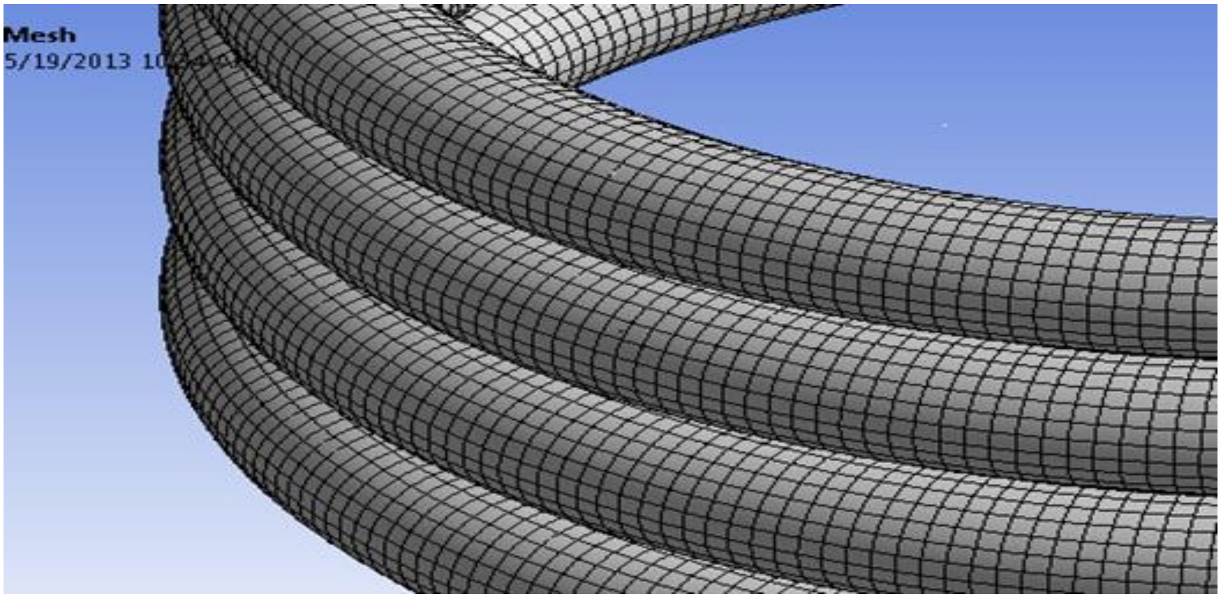

Fig.4.2 Grid of the computational domain

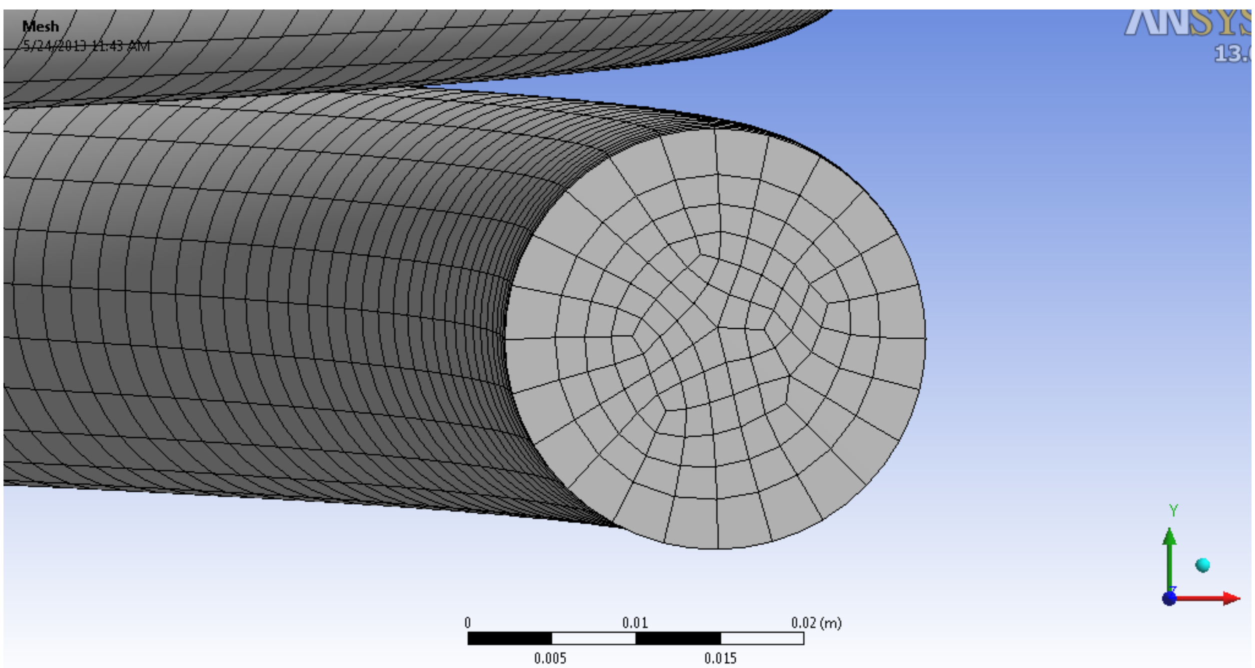

Fig.4.3 Front view of mesh

4.4.3 Flow Specification

The assumptions used in this model were

a. The flow was steady and incompressible. The fluid density was constant throughout the computational domain.
b. Water was the working fluid. The fluid properties (ρ, μ and specific heat) being constant throughout the computational domain.
c. The effect of heat conduction through the tube material is small.

For the present analysis the method applied is explained below. All the governing equations used in present analysis were solved by using ANSYS FLUENT 13.0 finite volume commercial code. Second order upwind scheme was used for solving momentum and energy equations. The convergence criterion was fixed such that the residual value was lower than 1e-6. The pressure correction approach using the SIMPLE algorithm was used. Relaxation factor have been kept to default values. Refer table 4.1 for values.

Table 4.1 Relaxation factors

Pressure	Momentum	Energy	Density	Body Force
0.3	0.7	1	1	1

Mass flow rate was given at the inlet whereas static pressure was given at outlet for velocity inlet and pressure outlet boundary condition and static pressure was given at the inlet as well as at the output for pressure inlet and pressure outlet boundary condition. The input parameters were indirectly taken from the Reynolds number value. Uniform heat flux was applied for the wall of the pipe under constant wall heat flux condition and uniform wall temperature was specified for constant wall temperature condition. The turbulence model applied for present analysis was k-epsilon model.

CHAPTER-5

RESULTS AND DISCUSSION

5.1 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.

5.1.1 Contours

Figures 5.2 and 5.3 shows pressure contour and temperature contour respectively for the boundary condition of constant wall temperature. The flow behavior is turbulent and inlet velocity is 0.6 m/s for this case.

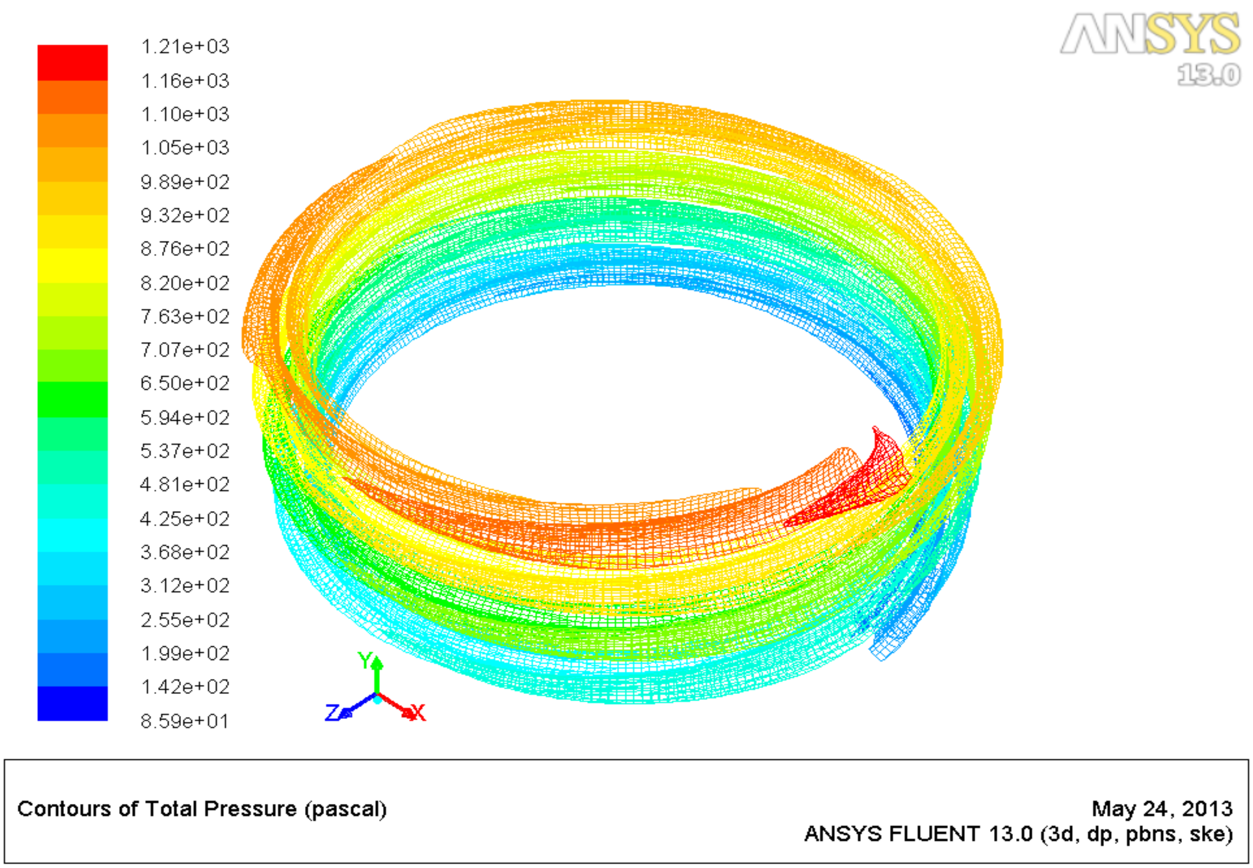

Fig.5.1 Contour of pressure distribution

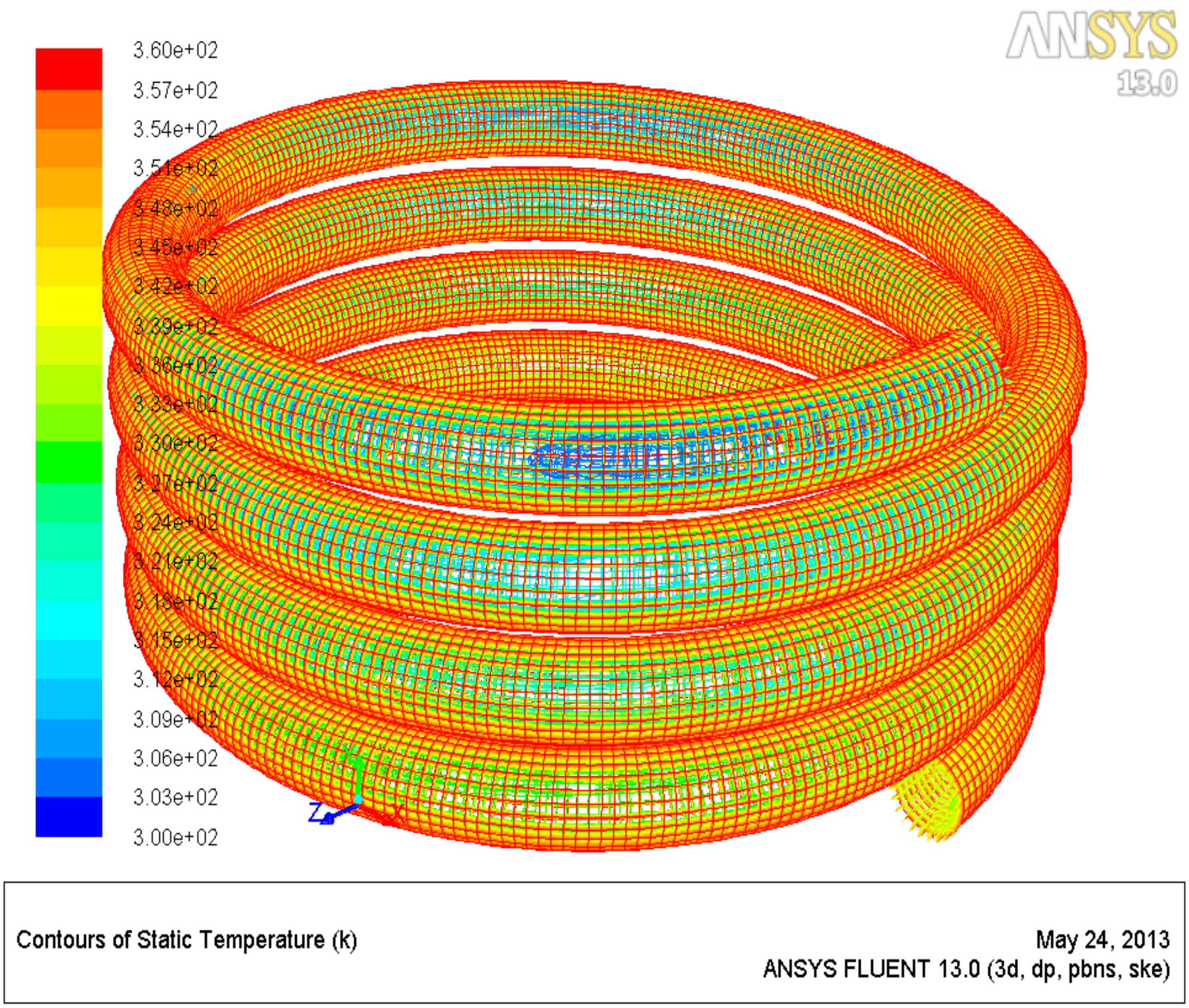

Fig.5.2 Contour of temperature distribution inside pipe

5.1.2 Tabulation

The results obtained from the CFD methodology have been used to calculate the values of Nusselt number (Nu) and friction factor (f) which has been tabulated in tables 5.1-5.4.

Table 5.1 Nu and f values for constant wall temperature and turbulent flow

D/d	V (m/s)	Nu	f
12	0.6	138.86	0.036
15	0.6	111.33	0.037
20	0.6	86.26	0.038
30	0.6	59.37	0.041
12	0.8	174.13	0.034
15	0.8	140.07	0.035
20	0.8	108.37	0.036
30	0.8	74.63	0.038
12	1	208.12	0.032
15	1	169.36	0.033
20	1	129.73	0.034
30	1	89.27	0.036
12	1.2	243.43	0.030
15	1.2	198.28	0.031
20	1.2	150.24	0.032
30	1.2	103.42	0.034

Table 5.2 Nu and f values for constant wall temperature and laminar flow

D/d	V (m/s)	Nu	f
12	0.6	18.57	0.0145
15	0.6	17.47	0.0159
20	0.6	15.73	0.0191
30	0.6	13.59	0.0209
12	0.8	18.58	0.0126
15	0.8	17.5	0.0128
20	0.8	15.75	0.0151
30	0.8	13.61	0.0165
12	1	18.62	0.0106
15	1	17.56	0.0110
20	1	15.75	0.0125
30	1	13.61	0.0139
12	1.2	18.64	0.0097
15	1.2	17.57	0.0099
20	1.2	15.81	0.0109
30	1.2	13.65	0.0121

Table 5.3 Nu and f values for constant wall heat flux and turbulent flow

D/d	V (m/s)	Nu	f
12	0.6	133.44	0.036
15	0.6	106.82	0.037
20	0.6	81.62	0.038
30	0.6	56.14	0.041
12	0.8	167.37	0.034
15	0.8	134.57	0.035
20	0.8	103.56	0.036
30	0.8	71.23	0.038
12	1	200.22	0.032
15	1	161.77	0.033
20	1	124.22	0.034
30	1	85.76	0.036
12	1.2	233.04	0.030
15	1.2	187.95	0.031
20	1.2	145.26	0.032
30	1.2	99.86	0.034

Table 5.4 Nu and f values for constant wall heat flux and laminar flow

D/d	V (m/s)	Nu	f
12	0.6	18.47	0.0145
15	0.6	17.40	0.0159
20	0.6	15.63	0.0191
30	0.6	13.15	0.0209
12	0.8	18.49	0.0126
15	0.8	17.45	0.0128
20	0.8	15.68	0.0151
30	0.8	13.20	0.0165
12	1	18.54	0.0106
15	1	17.51	0.0110
20	1	15.74	0.0125
30	1	13.26	0.0139
12	1.2	18.57	0.0097
15	1.2	17.53	0.0099
20	1.2	15.76	0.0109
30	1.2	13.28	0.0121

5.1.3 Graphs

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

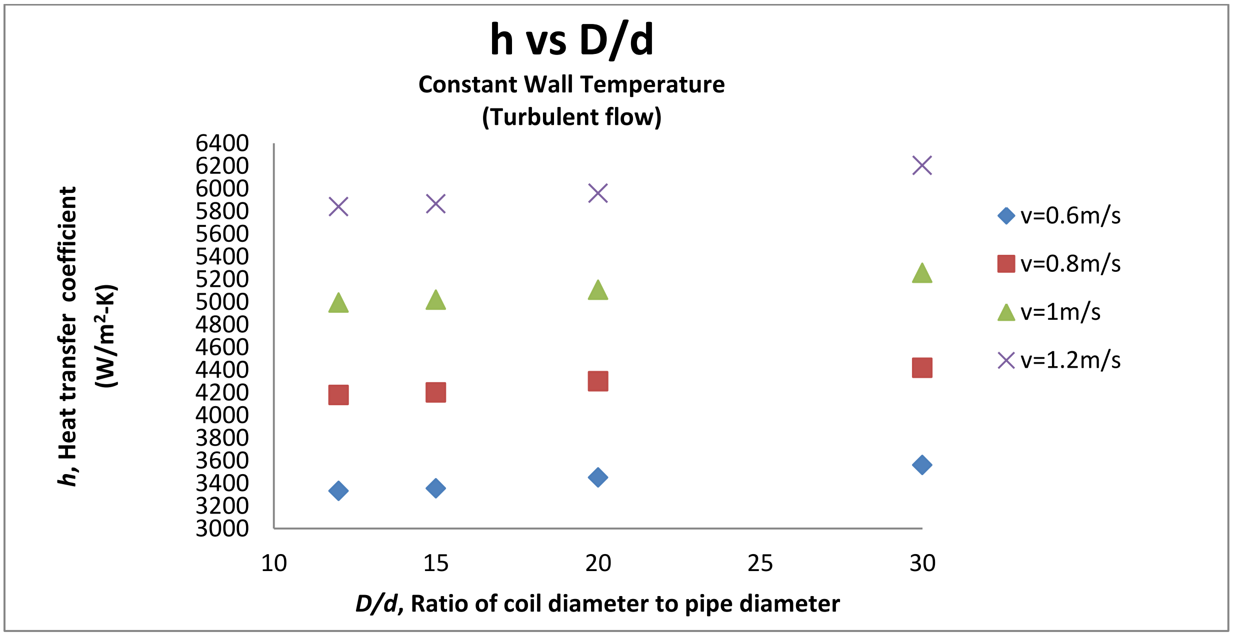

Fig.5.3 h vs. D/d for constant wall temperature (turbulent flow)

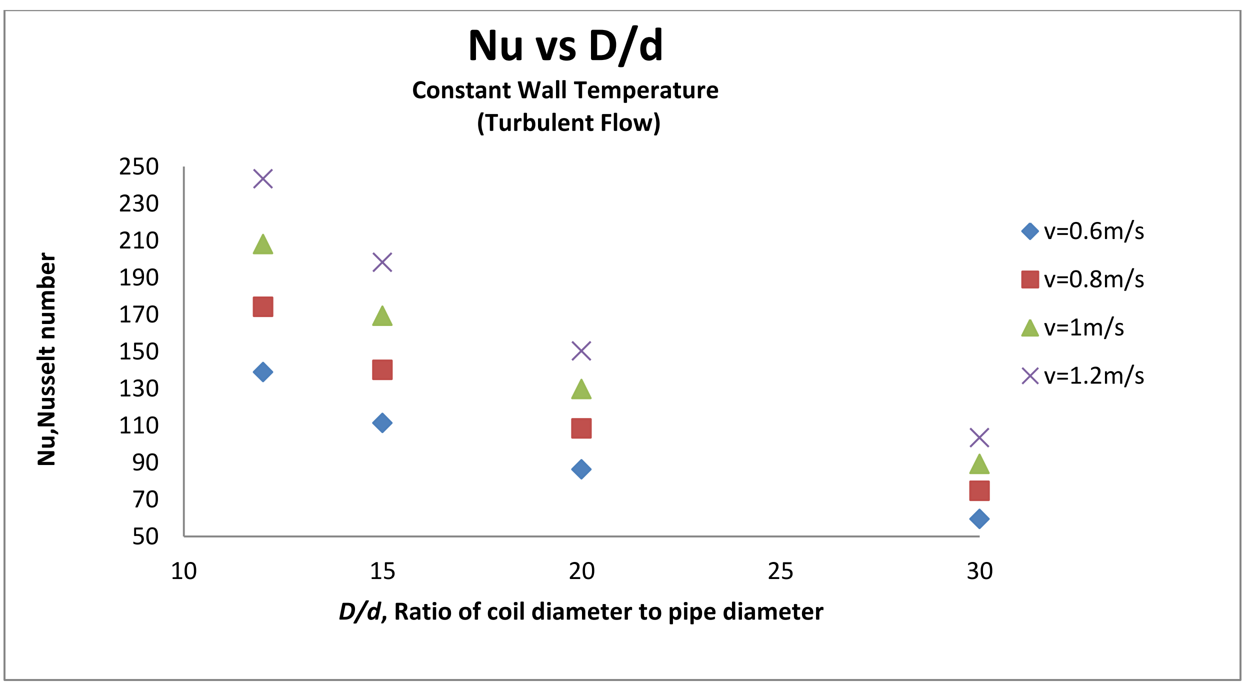

Fig.5.4 Nu vs. D/d for constant wall temperature (turbulent flow)

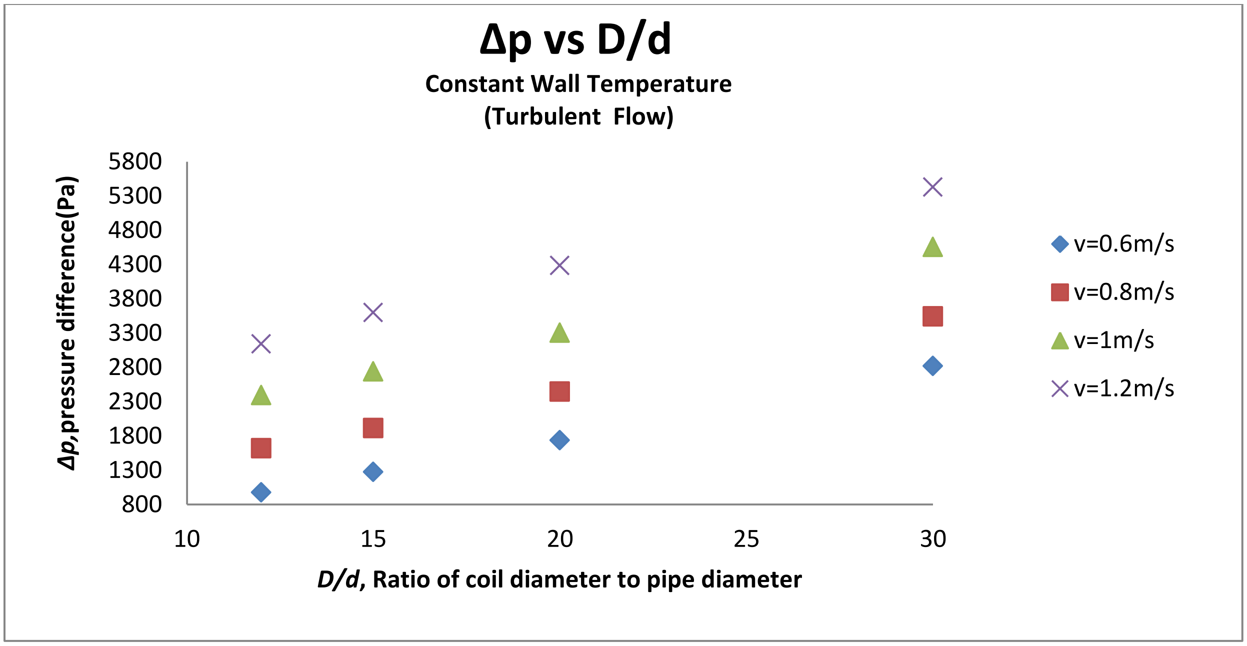

Fig.5.5 Δp vs. D/d for constant wall temperature (turbulent flow)

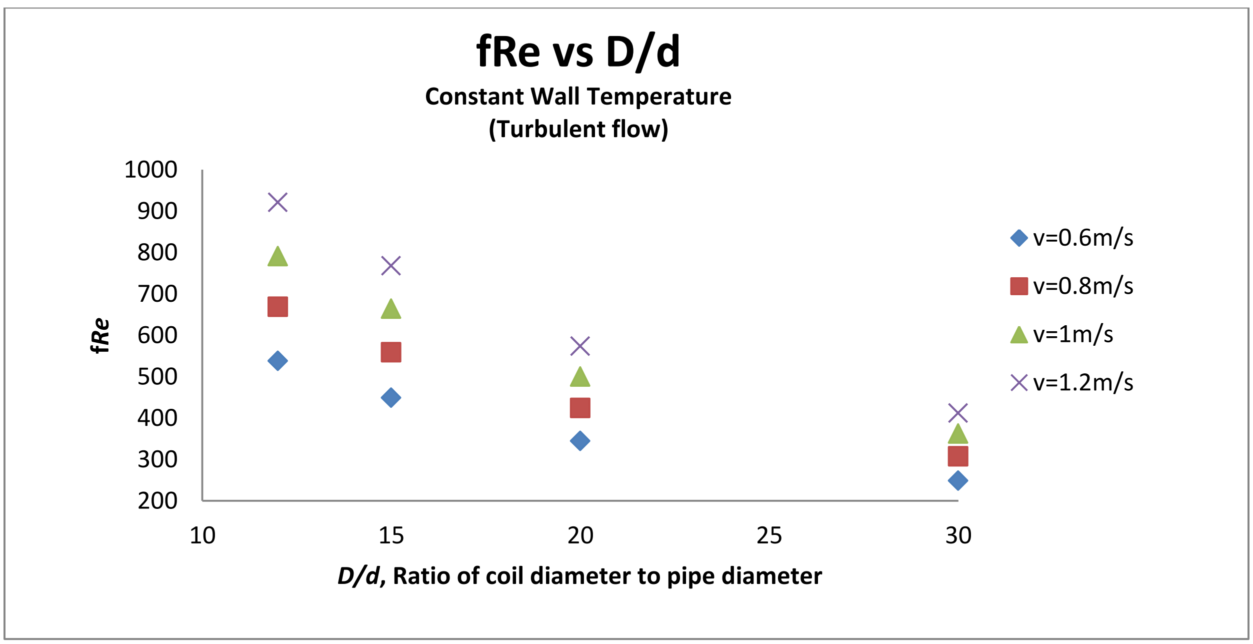

Fig.5.6 fRe vs. D/d for constant wall temperature (turbulent flow)

Figures 5.3-5.6 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 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.

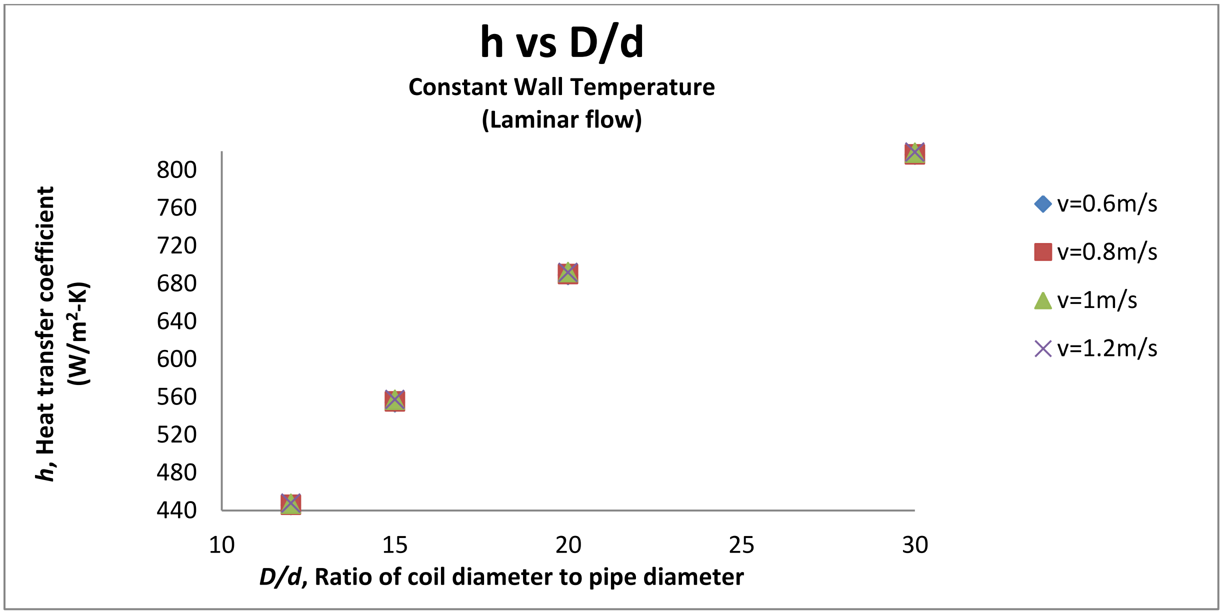

Fig.5.7 h vs. D/d for constant wall temperature (laminar flow)

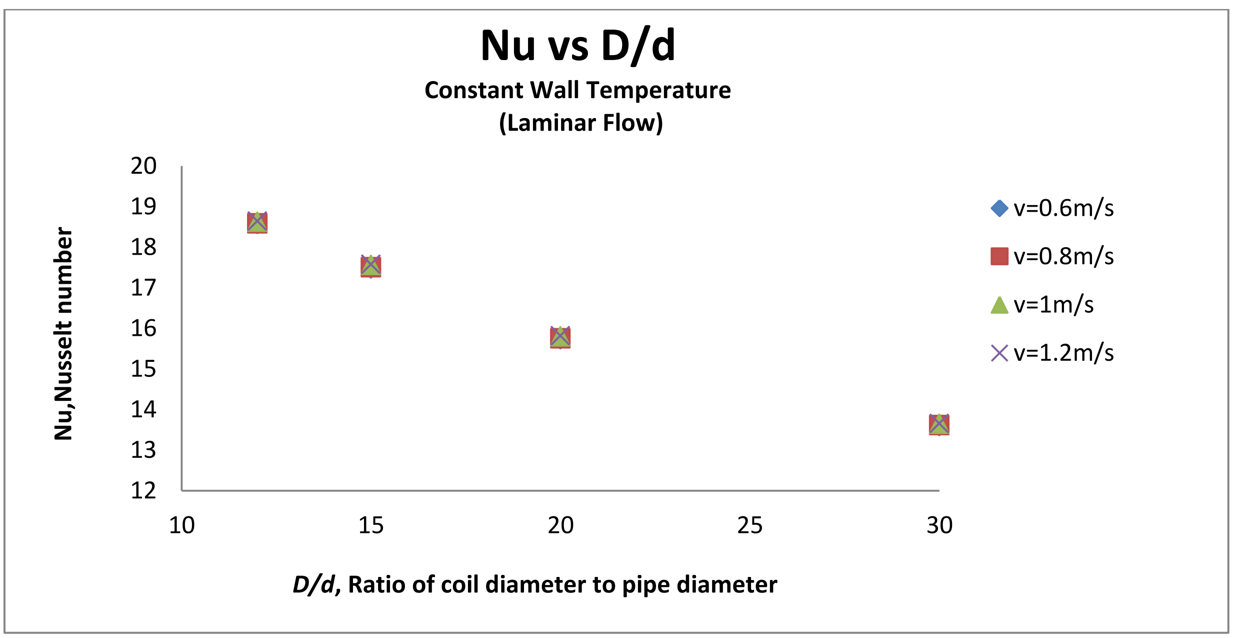

Fig.5.8 Nu vs. D/d for constant wall temperature (laminar flow)

From the figures it is also clear that as the inlet velocity or in other words mass flow rate is increasing, Nusselt number and other parameters are increasing which corresponds with the theory.

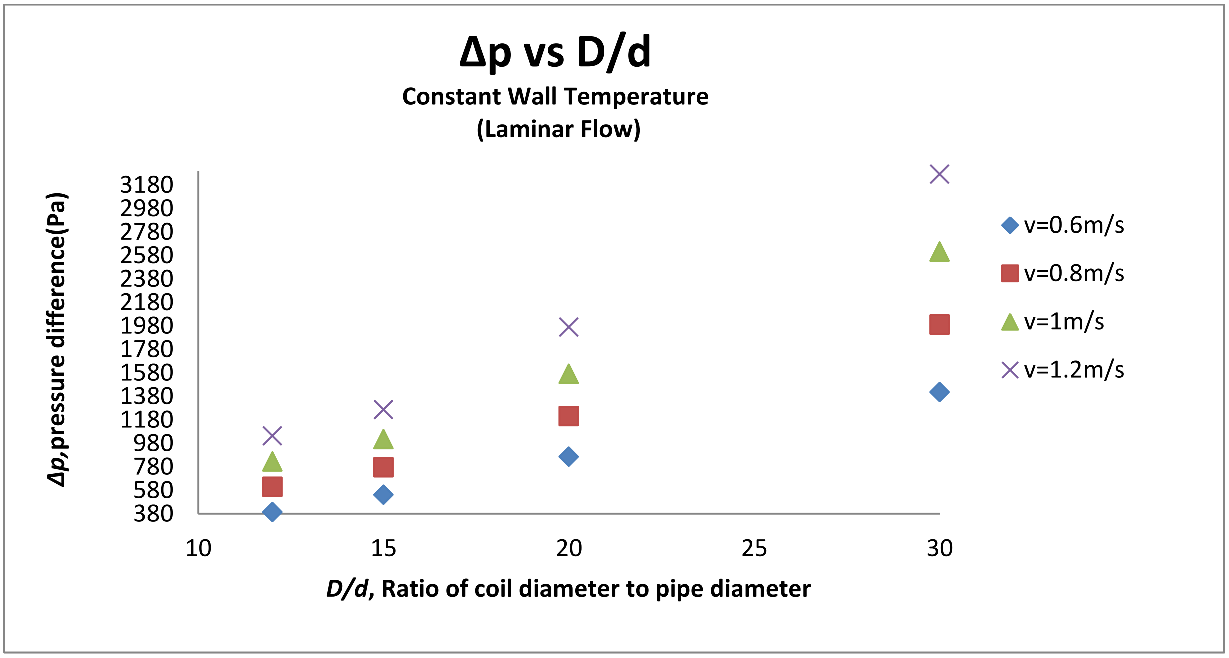

Fig.5.9 Δp vs. D/d for constant wall temperature (laminar flow)

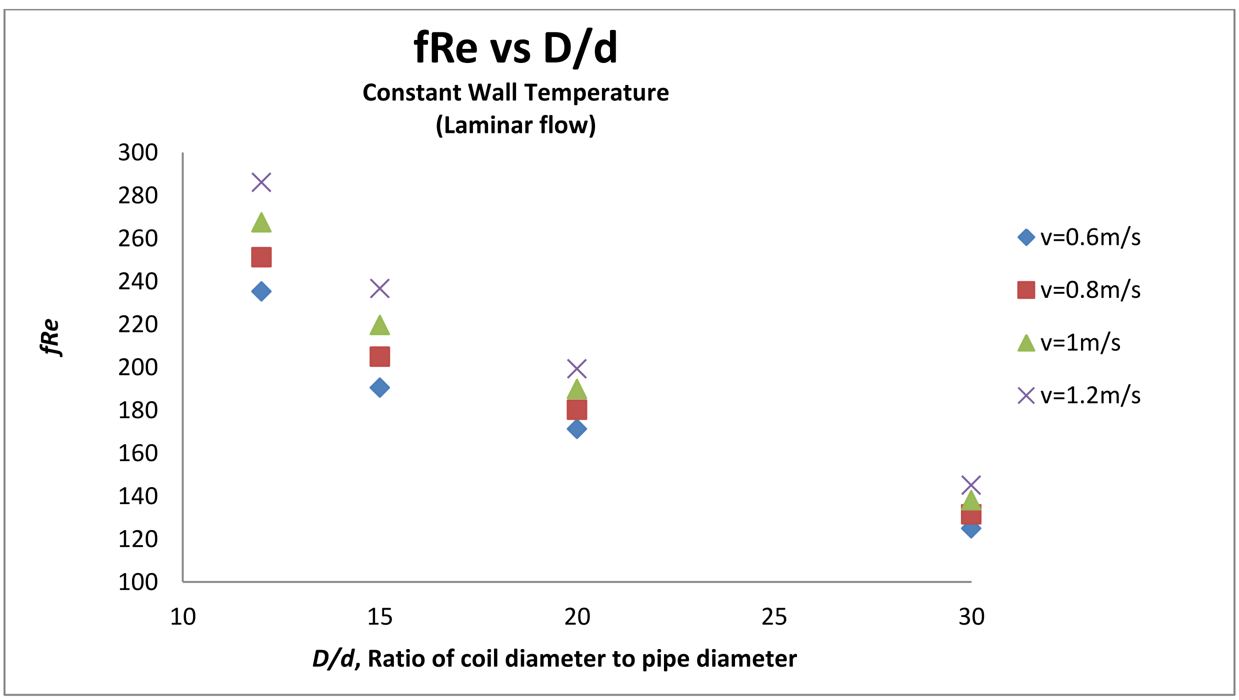

Fig.5.10 fRe vs. D/d for constant wall temperature (laminar flow)

Figure 5.7-5.10 shows heat transfer and fluid flow characteristics for constant wall temperature and laminar flow. In this case Nusselt number varies slightly with mass flow rate i.e. there will be a marginal change in value of Nusselt number with increase in inlet velocity. However the dependence of heat transfer and fluid flow characteristics on curvature ratio is same as that in the case of turbulent flow. Also in case of laminar flow, values of fRe and Nusselt number are much less than that in case of turbulent flow. So, we can also say that for better performance of a helical pipe, flow should be turbulent.

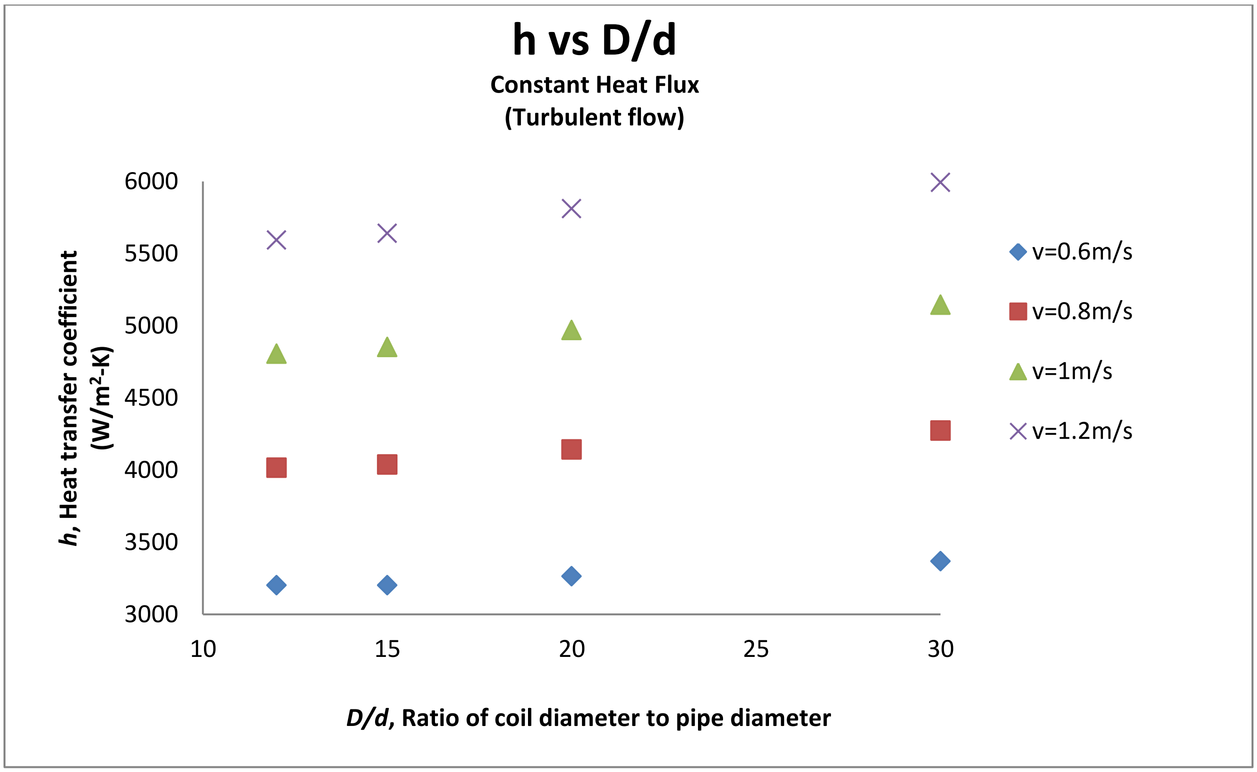

Fig. 5.11 h vs. D/d for constant wall heat flux (turbulent flow)

We can easily analyze the heat transfer and fluid flow characteristics of helical pipe for constant wall heat flux boundary condition from figures 5.11-5.14. The flow behavior is turbulent while figures 5.15-5.18 are also for same boundary condition but the flow behavior is laminar for these graphs. The heat transfer and fluid flow characteristics are similar to constant wall temperature boundary condition. But for the equivalent values of wall temperature and wall heat flux, Nusselt number will be slightly higher in constant wall heat flux boundary condition as reported by Jayakumar et al. (2008).

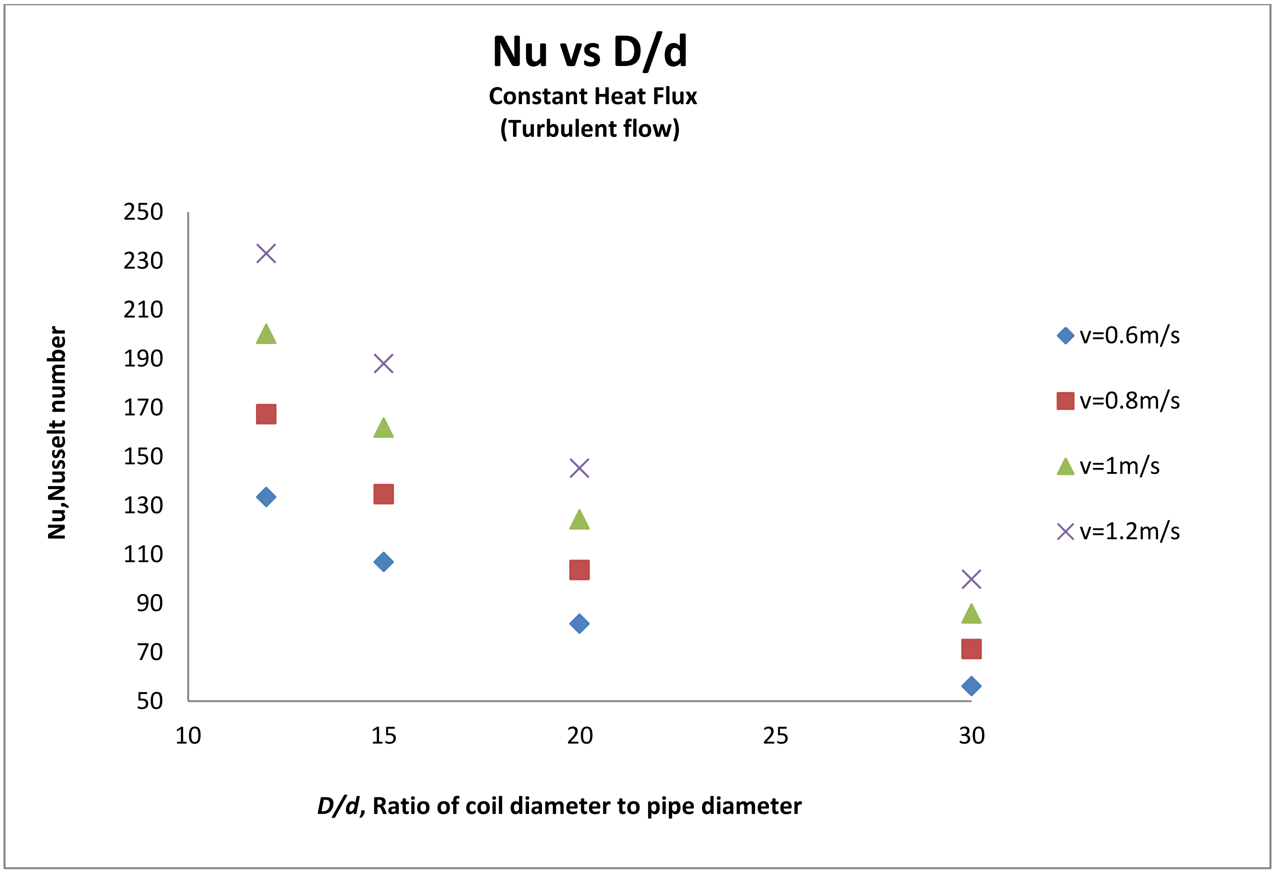

Fig.5.12 Nu vs. D/d for constant wall heat flux (turbulent flow)

As can be seen from figures5.6, 5.10, 5.14 and 5.18 that fRe varies with curvature ratio as well as mass flow rate and it increases with increase in curvature ratio, so we can analyse that there must be a limit to curvature ratio beyond which performance of helical pipe will deteriorate. Pressure difference is decreasing with increase in curvature ratio as visible from figures 5.5, 5.9, 5.13 and 5.17 which also favours our prediction that flow and heat transfer characteristics will improve with increase in curvature ratio.

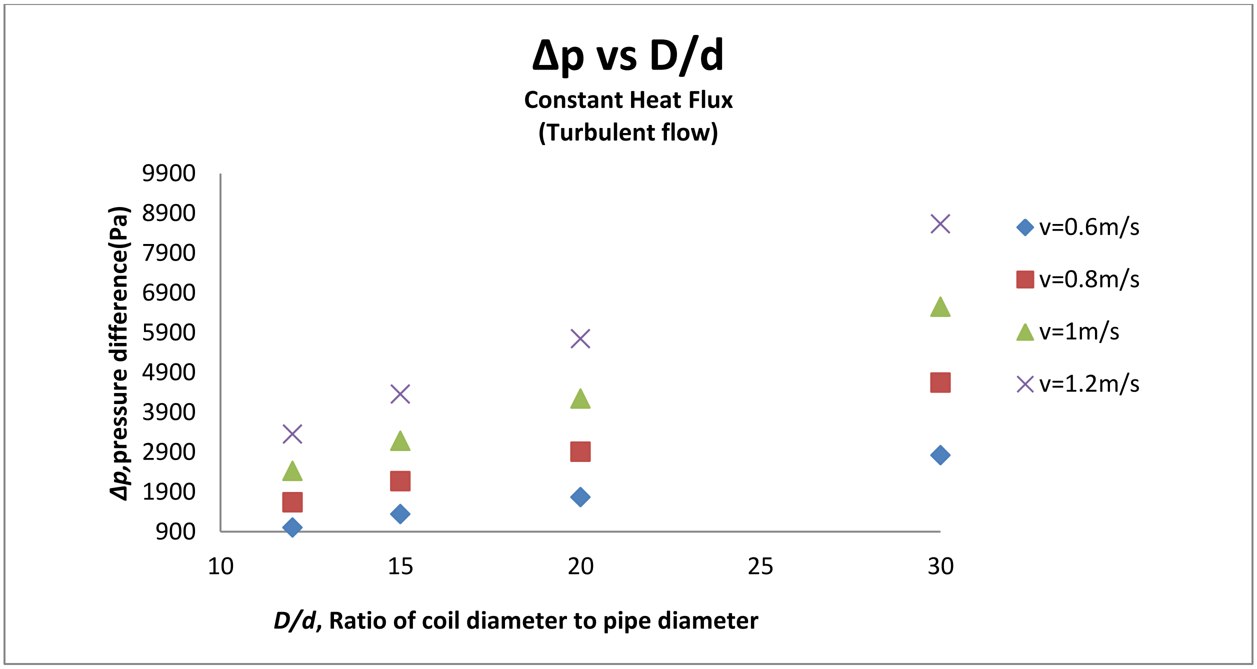

Fig.5.13 Δp vs. D/d for constant wall heat flux (turbulent flow)

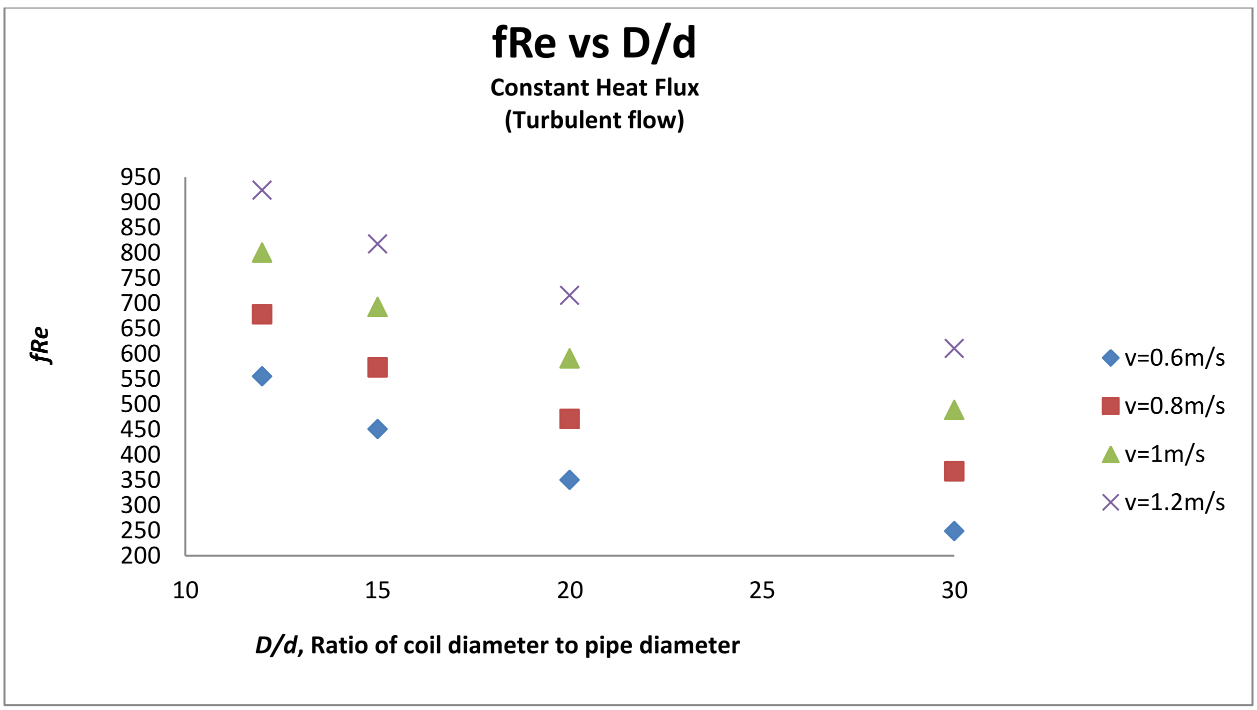

Fig.5.14 fRe vs. D/d for constant wall heat flux (turbulent flow)

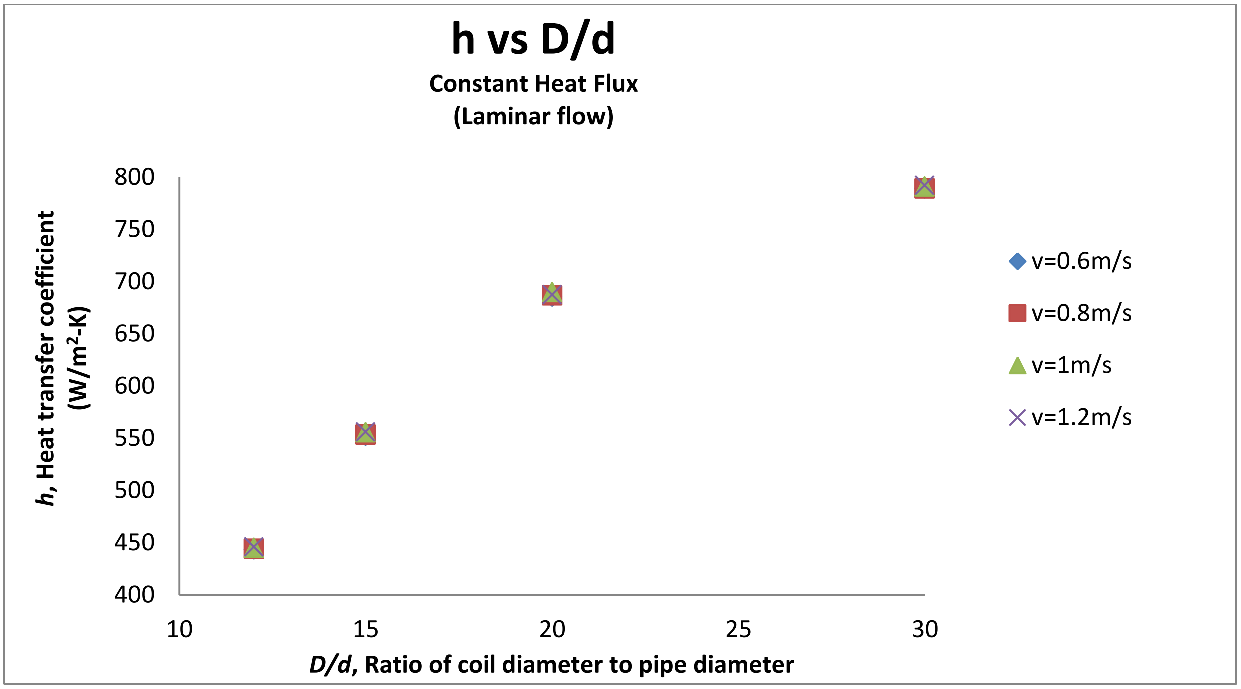

Fig.5.15 h vs. D/d for constant wall heat flux (laminar flow)

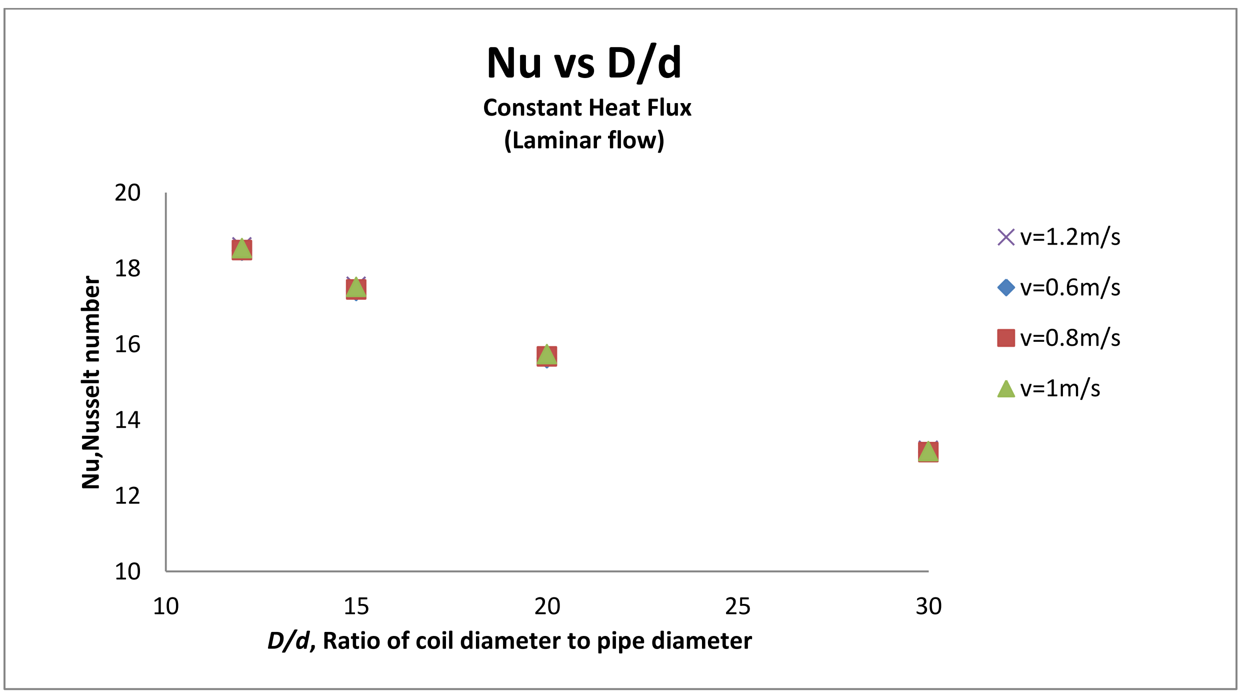

Fig.5.16 Nu vs. D/d for constant wall heat flux (laminar flow)

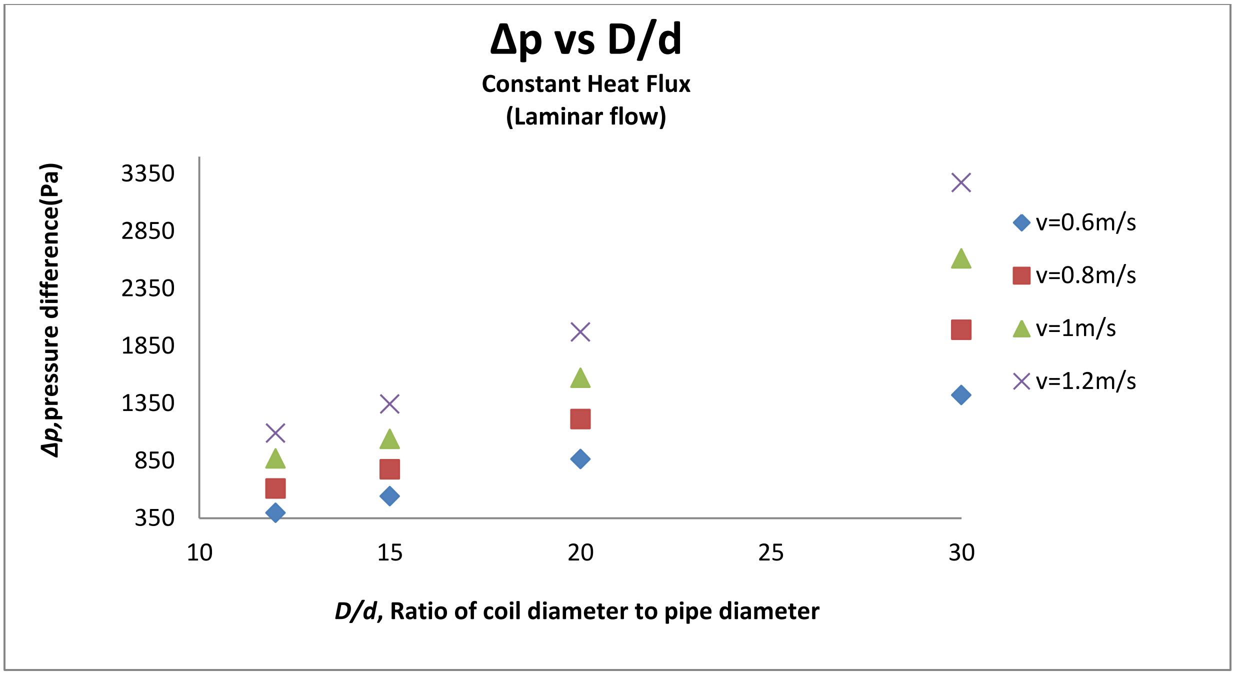

Fig. 5.17 Δp vs. D/d for constant wall heat flux (laminar flow)

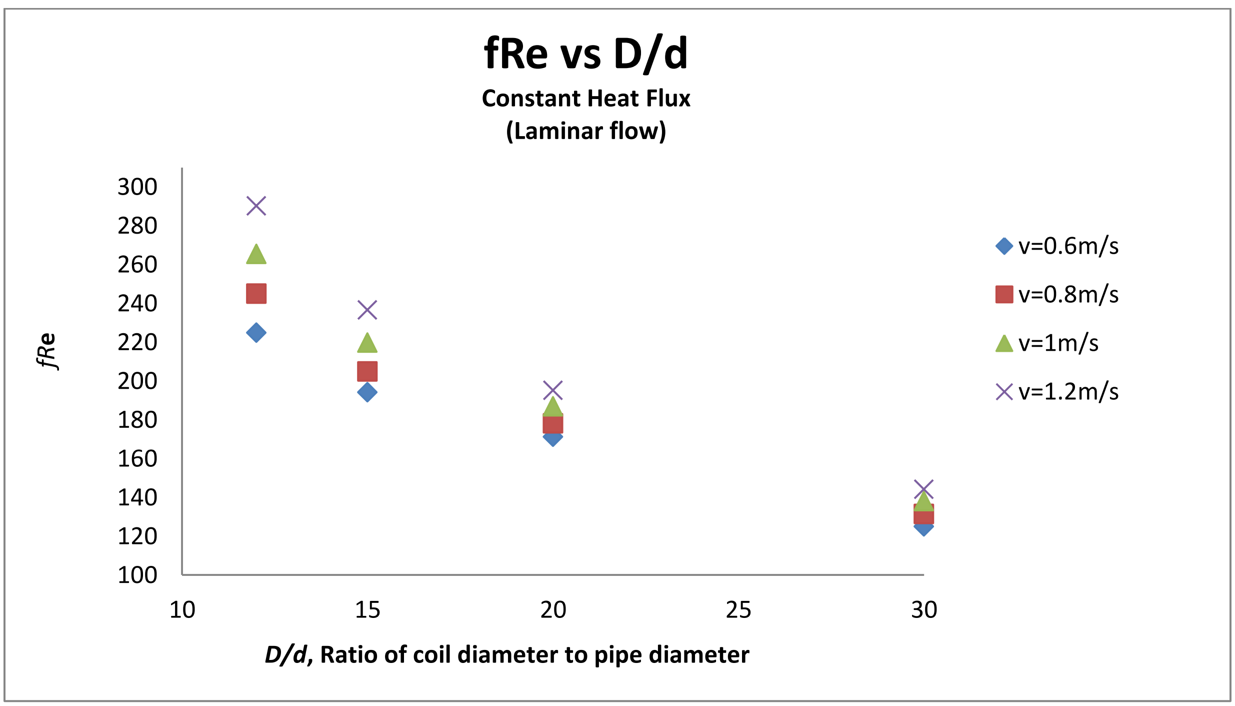

Fig.5.18 fRe vs. D/d for constant wall heat flux (laminar flow)

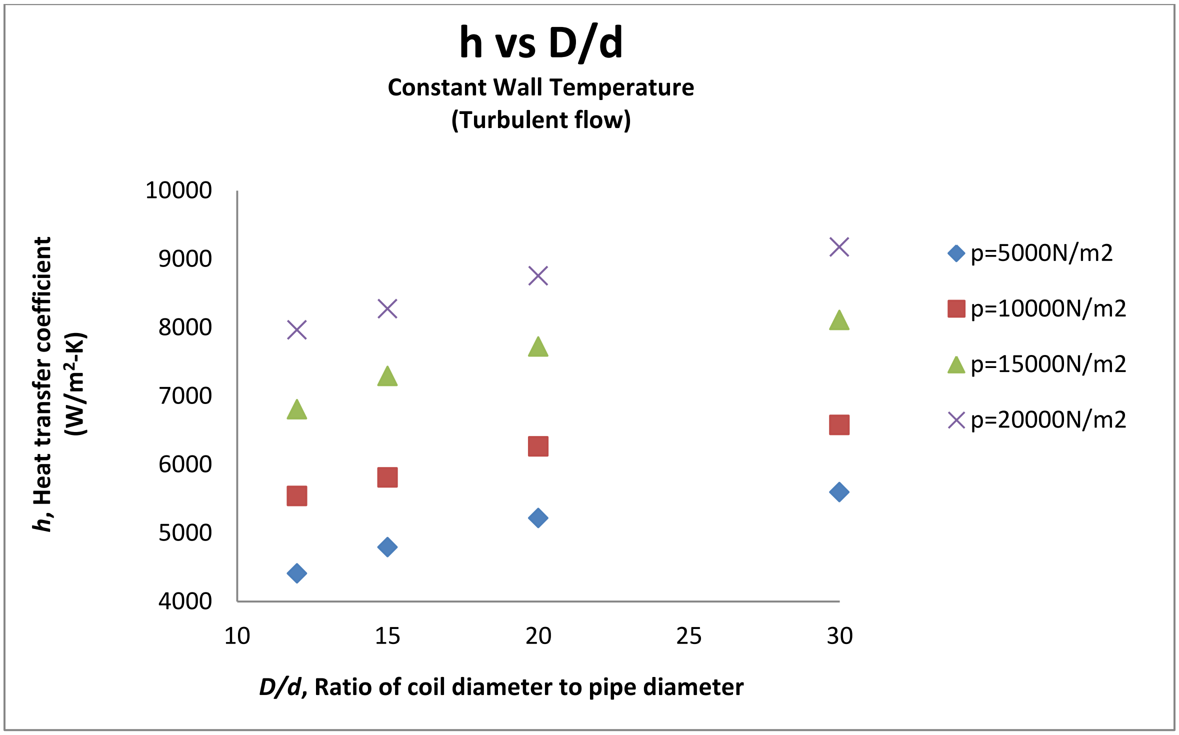

Fig.5.19 h vs. D/d for constant wall temperature (turbulent flow) and pressure inlet

Figures 5.19-5.21 shows variation of heat transfer coefficient, Nusselt number and pressure drop with D/d respectively for constant wall temperature and turbulent flow under variable inlet pressure. As the inlet pressure is increasing, Nusselt number is also increasing with increase in curvature ratio or one can say with decrease in D/d ratio (ratio of coil diameter to pipe diameter). So, these results also confirm our analysis that a helical pipe will give better performance with increase in curvature ratio. From figure 5.19 it is clear that heat transfer coefficient (h) is increasing with increase in inlet pressure.

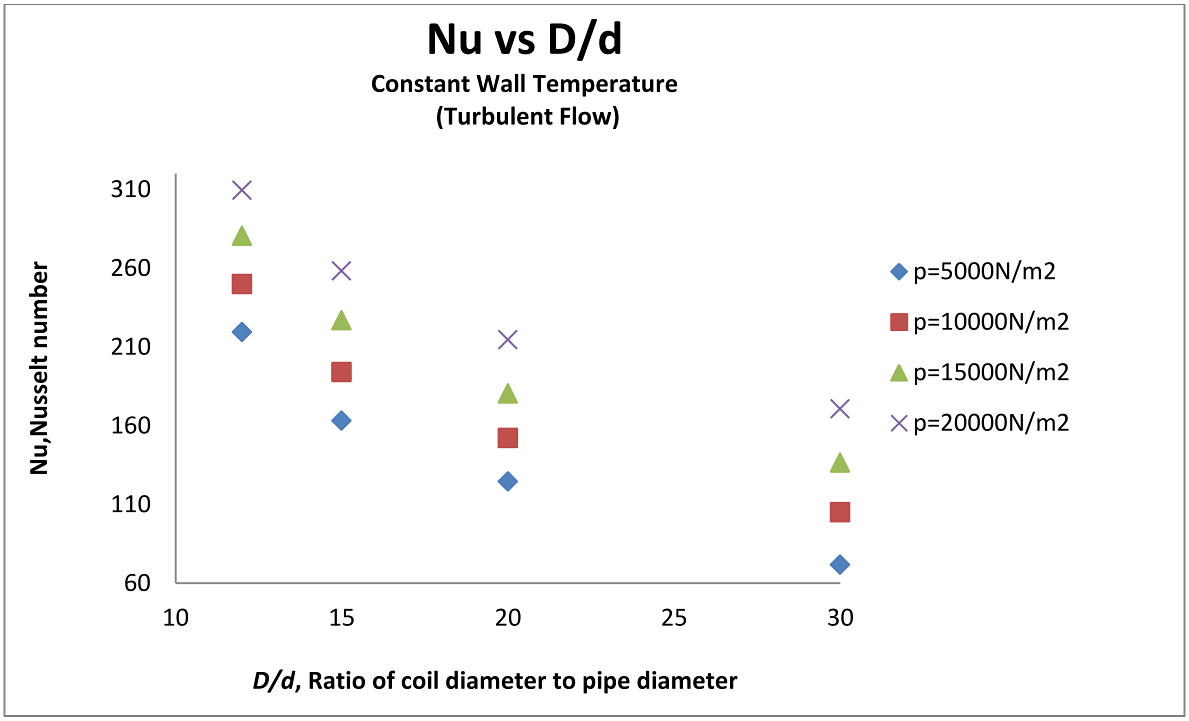

Fig.5.20 Nu vs. D/d for constant wall temperature (turbulent flow) and pressure inlet

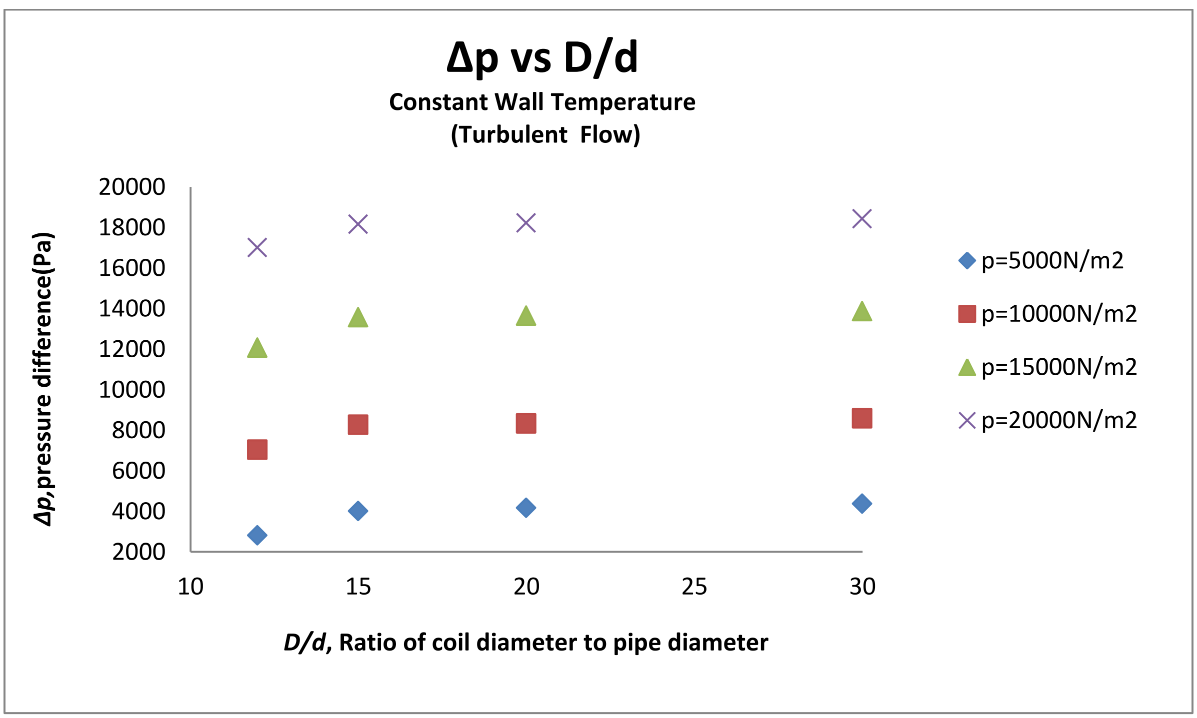

Fig.5.21 Δp vs. D/d for constant wall temperature (turbulent flow) and pressure inlet

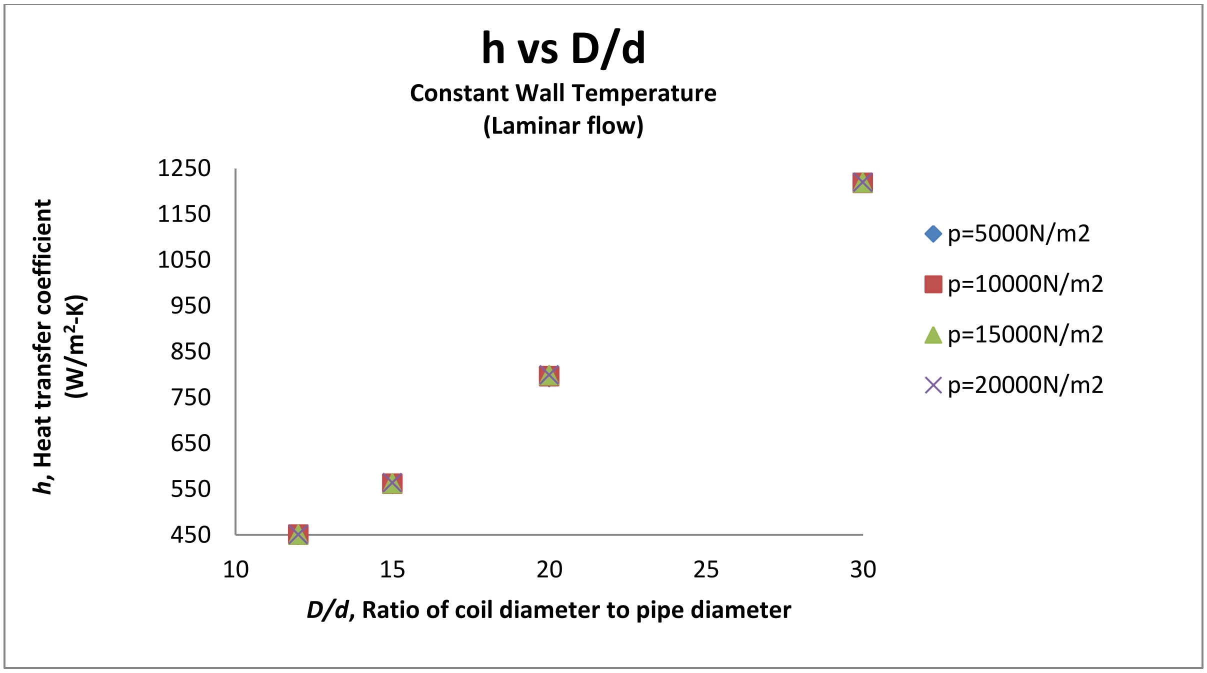

Fig.5.22 h vs. D/d for constant wall temperature (laminar flow) and pressure inlet

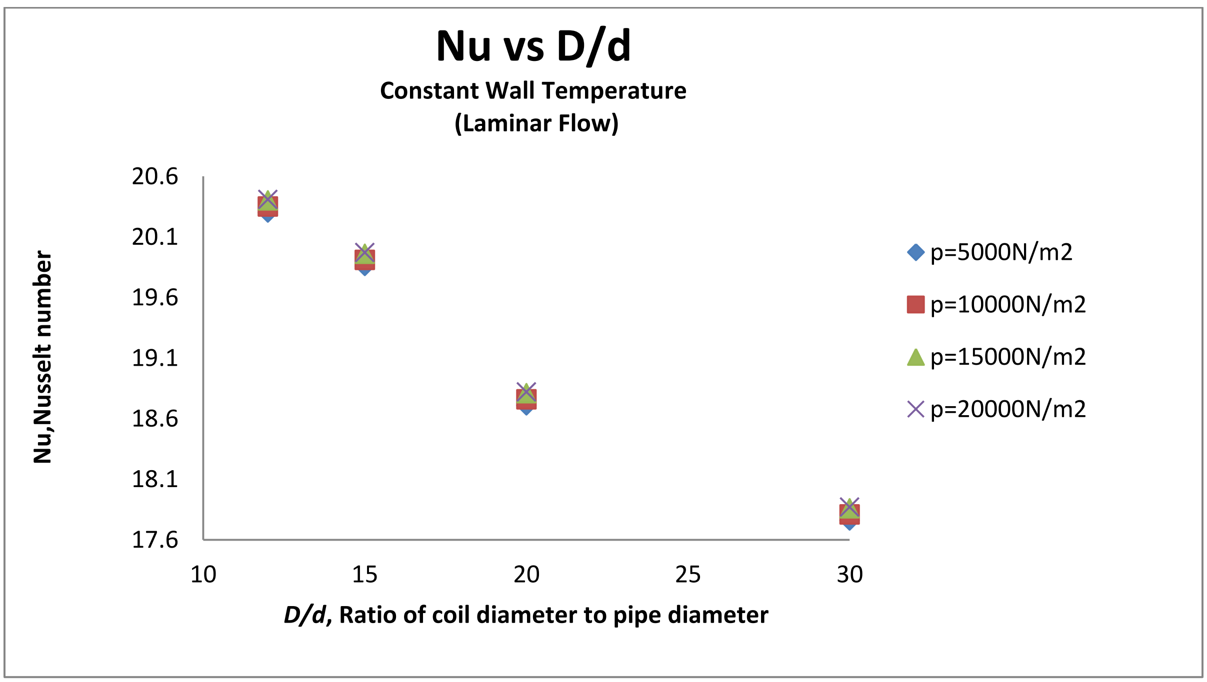

Fig.5.23 Nu vs. D/d for constant wall temperature (laminar flow) and pressure inlet

Figures 5.22-5.24 shows variation of heat transfer coefficient (h), Nusselt number (Nu) and pressure drop (Δp) with D/d ratio respectively for laminar flow and constant wall temperature. As we have seen before that Nusselt number remains almost constant with slight change in mass flow rate in case of laminar flow, similarly here also it is almost constant with change in inlet pressure.

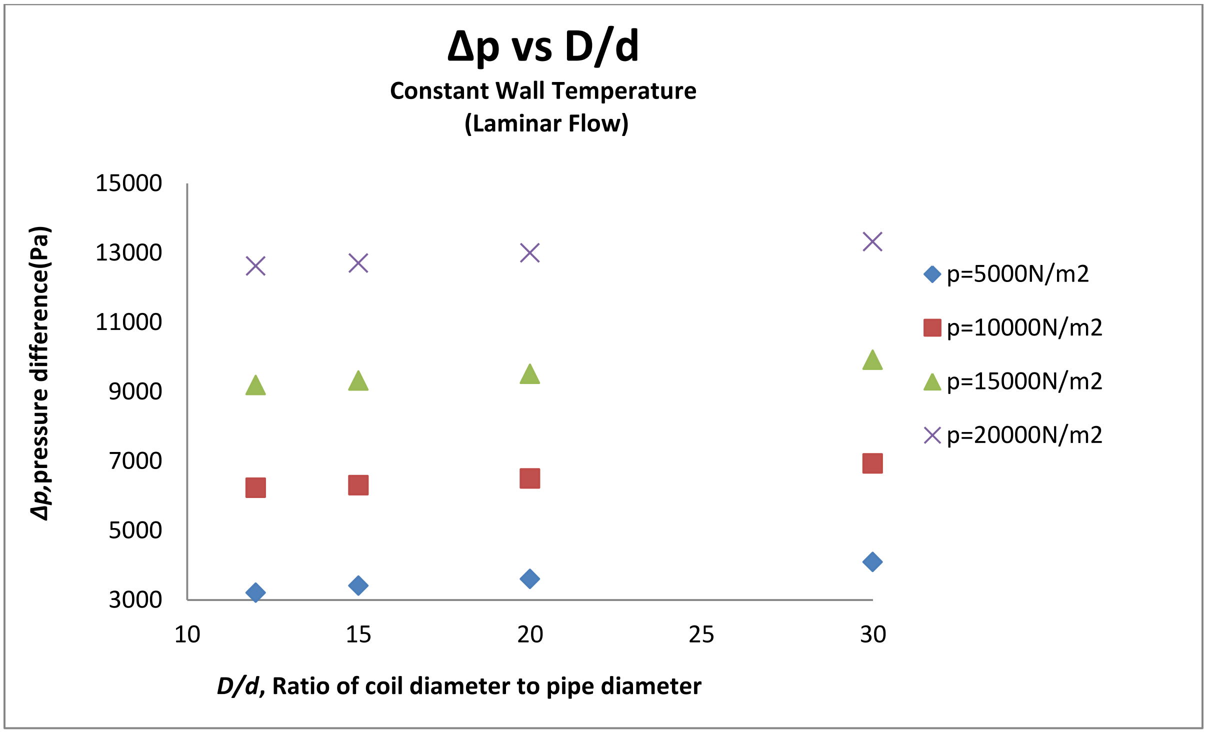

Fig.5.24 Δp vs. D/d for constant wall temperature (laminar flow) and pressure inlet

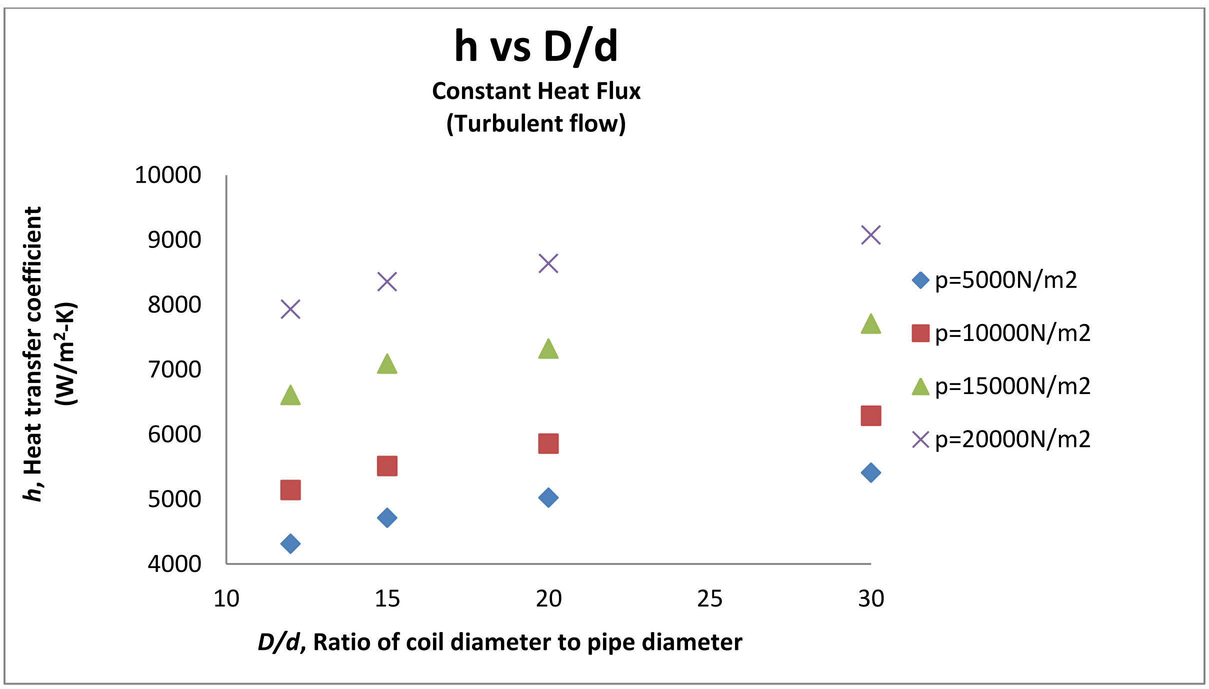

Fig.5.25 h vs. D/d for constant wall heat flux (turbulent flow) and pressure inlet

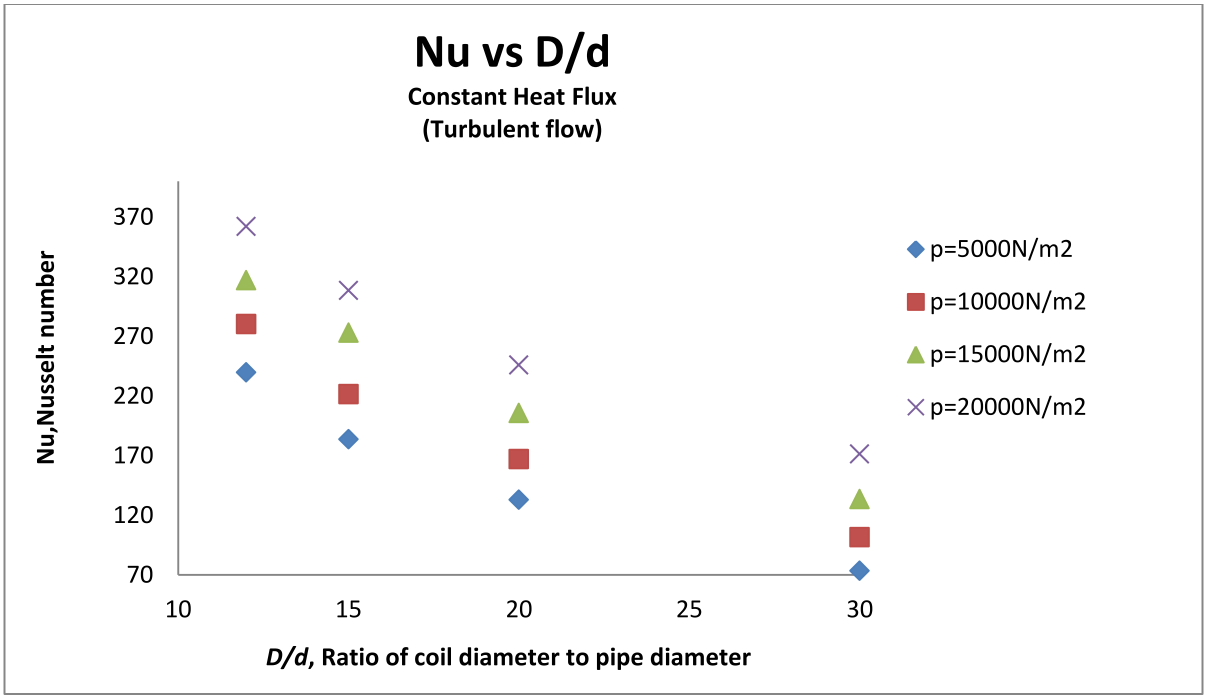

Fig.5.26 Nu vs. D/d for constant wall heat flux (turbulent flow) and pressure inlet

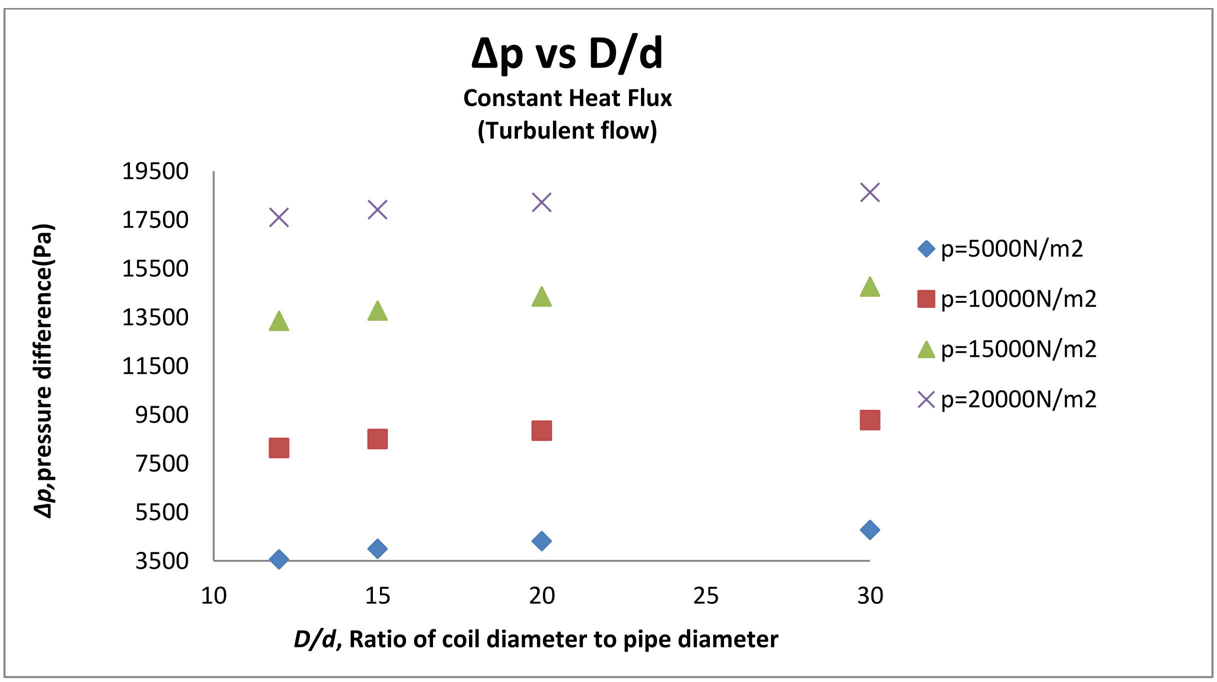

Fig.5.27 Δp vs. D/d for constant wall heat flux (turbulent flow) and pressure inlet

From figures 5.25-5.27 we can visualize the heat transfer and fluid flow characteristics for constant wall heat flux boundary condition and turbulent flow under variable inlet pressure.

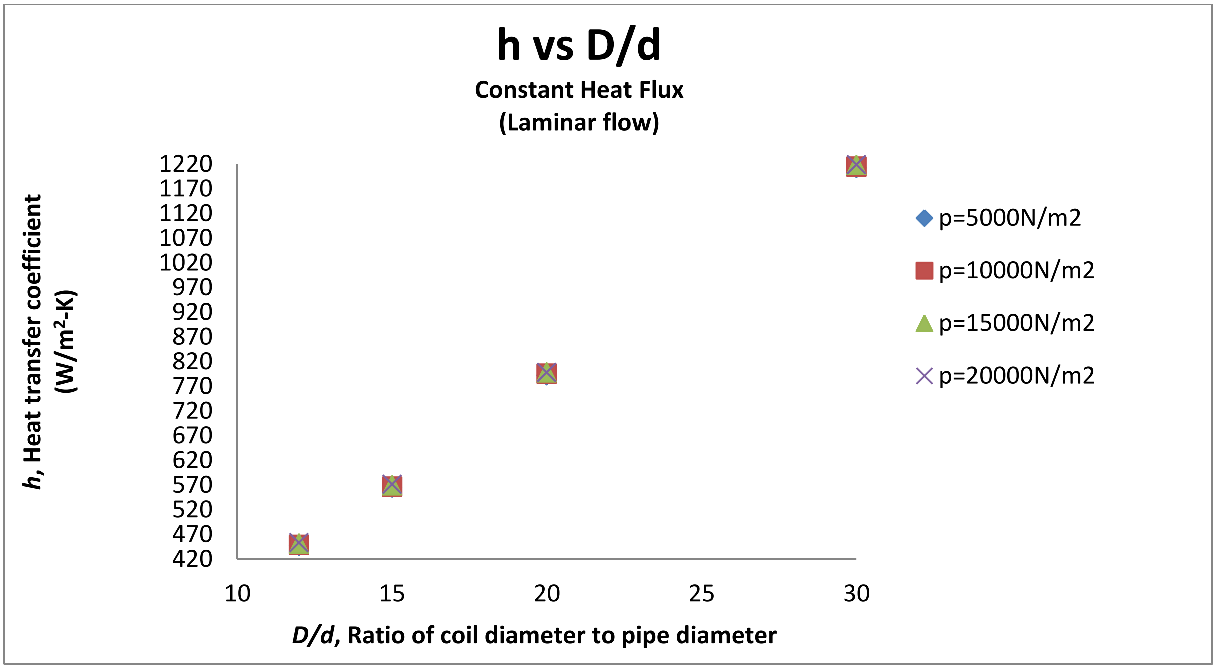

Fig.5.28 h vs. D/d for constant wall heat flux (laminar flow) and pressure inlet

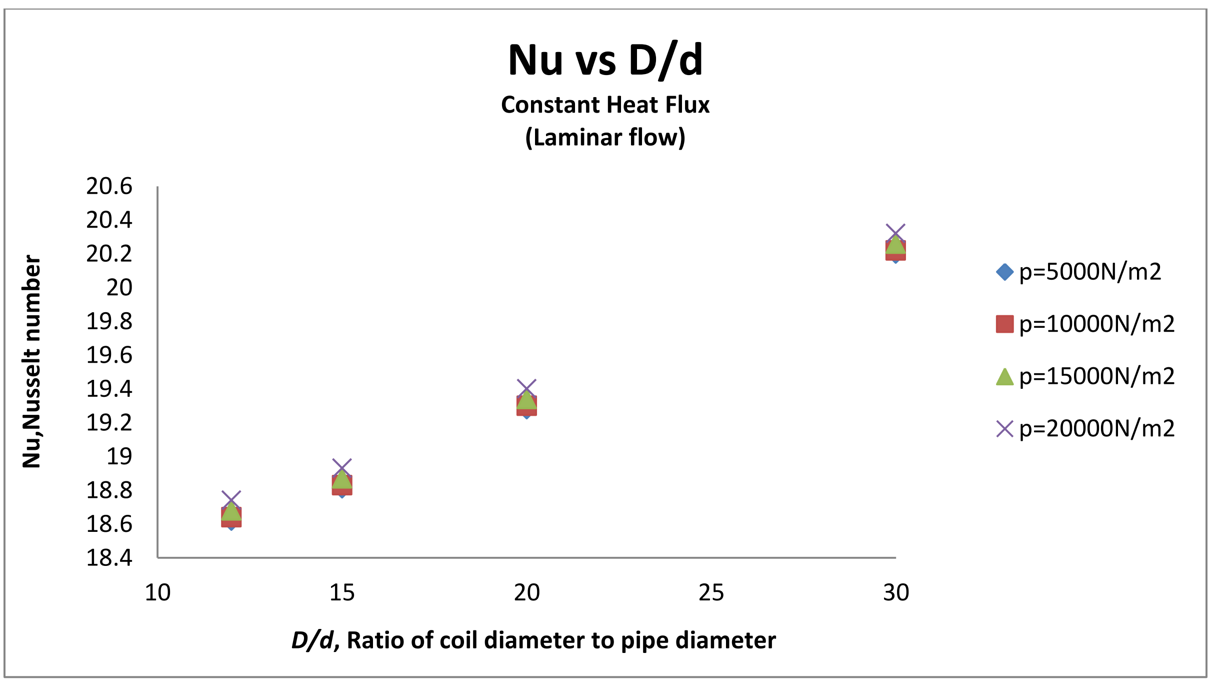

Fig.5.29 Nu vs. D/d for constant wall heat flux (laminar flow) and pressure inlet

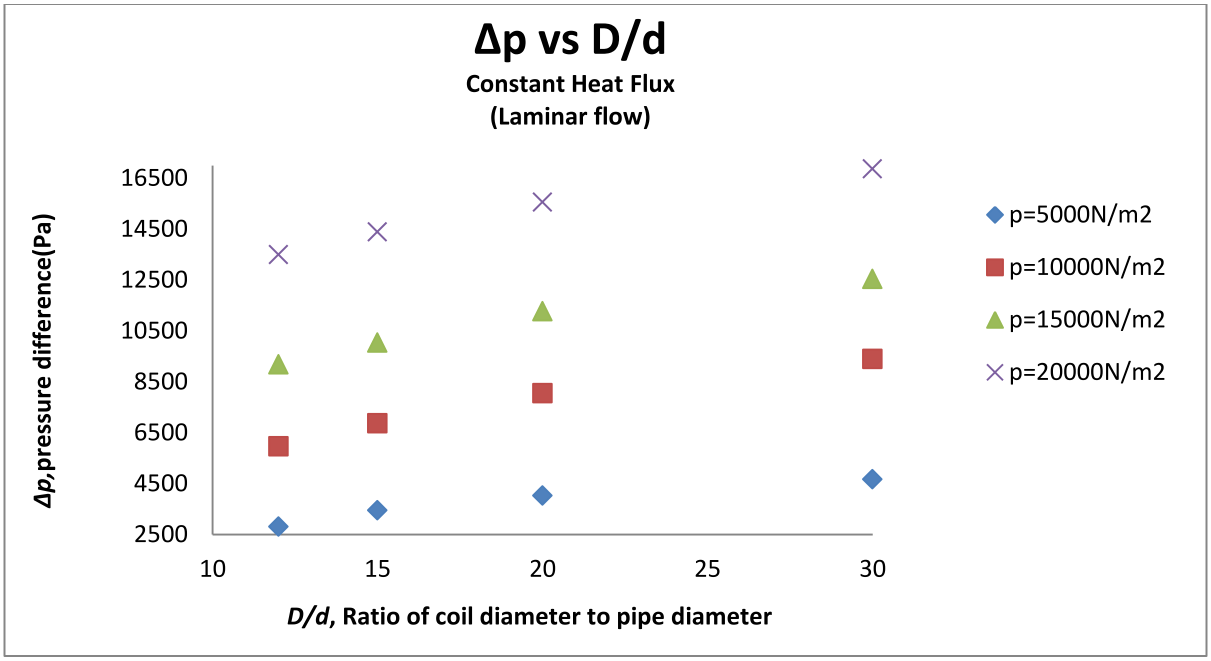

Fig.5.30 Δp vs. D/d for constant wall heat flux (laminar flow) and pressure inlet

The heat transfer and fluid flow characteristics of a helical pipe with varying inlet pressure for constant wall heat flux and laminar flow can be observed from figures 5.28-5.30. The behavior of the above plotted parameters is similar to that in the case of constant wall temperature. Jayakumar et al. (2008) also did CFD analysis for helical pipe for various wall boundary conditions and turbulent flow. Results obtained here correspond with that of them. So, results could be validated.

Also from the CFD analysis we have found that Nusselt number of a helical pipe is higher than that of a straight pipe which corresponds with the theory and experimental results which give confidence about our CFD methodology.

CHAPTER-6

CONCLUSIONS AND FUTURE SCOPE

6.1 Conclusions

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.

6.2 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.

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. Chem Eng Sci, 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.

[9] Ferng, Y.M., Lin, W.C., Chieng, C.C. Numerically investigated effects of different Dean number and pitch size on flow and heat transfer characteristics in a helically coil-tube heat exchanger, Applied Thermal Engineering, 36 (2012): 378-385.

[10] Grijspeerdt, K., et al. Application of computational fluid dynamics to model the hydrodynamics of plate type heat exchangers for milk processing. J Food Eng, 57 (2003): 237–242.

[11] Hesselgreaves J.E. , Compact Heat Exchangers: Selection, Design and Operation. Gulf Professional, 2001

[12] Janssen, L.A.M. and Hoogendoorn, C.J. Laminar convective heat transfer in helical coiled tubes. Int J Heat Mass Transf, 21 (1978): 1197–1206.

[13] Jayakumar, J.S. and Grover, R.B. Two phase natural circulation residual heat removal, In Proc. 3rd ISHMT-ASME Heat and Mass Transfer Conference, Kanpur, India. 1997

[14] Jayakumar, J.S., Grover, R.B. and Arakeri, V.H. Response of a two phase system subject to oscillations induced by the motion of its support structure. Int Comm Heat Mass Transf, 29 (2002): 519–530.

[15] Jayakumar, J.S., Mahajania, S.M., Mandal, J.C., Vijayan, P.K., Bhoia, Rohidas. Experimental and CFD estimation of heat transfer in helically coiled heat exchangers. Int J Chemical Engineering Research and Design. 86 (2008):221-232

[16] Jensen, M.K. and Bergles, A.E. Trans ASME, 103 (1981): 660– 666.

[17] Kalb, C.E. and Seader, J.D. Heat and Mass transfer phenomena for viscous flow in curved circular tubes. Int J Heat Mass Transf, 15 (1972): 801–817.

[18] Kang, H.J., Lin, C.X. and Ebadian, M.A. Condensation of R134a Flowing inside helicoidal pipe. Int J Heat Mass Transf, 43 (2000): 2553–2564.

[19] Kubair, V. and Kuloor, N.R. Heat transfer to Newtonian fluids in coiled pipes in laminar flow. Int J Heat Mass Transf, 9 (1996): 63–75.

[20] Kumar, V., Saini, S., Sharma, M. and Nigam, K.D.P. Pressure drop and heat transfer in tube in tube helical heat exchanger. Chem Eng Sci, 61 (2006): 4403–4416.

[21] Manna, R., Jayakumar, J.S. and Grover, R.B. Thermal Hydraulic design of a condenser for a natural circulation system. J Energy Heat Mass Transf, 18 (1996): 39–46.

[22] Mori, Y. and Nakayama. Study on forced convective heat transfer in curved pipes (3rd report). Int J Heat Mass Transf, 10 (1967): 681–695.

[23] Mori, Y. and Nakayama. Study on forced convective heat transfer in curved pipes (2nd report). Int J Heat Mass Transf, 10 (1967): 37 -59.

[24] Nandakumar, K. and Masliyah, J.H. Bifurcation in steady laminar flow through curved tubes. J Fluid Mech, 119 (1982): 475– 490.

[25] Naphon, P. and Wongwises, S. A review of flow and heat transfer characteristics in curved tubes. Renewable Sustain Energy Rev, 10 (2006): 463–490.

[26] Patankar, S., Pratap, V.S. and Spakling, D.B. Prediction of laminar flow and heat transfer in helically coiled pipes. J Fluid Mech, 62 (1974):539–551.

[27] Prabhanjan, D.G., Rennie, T.J. and Raghavan, G.S.V. Natural convection heat transfer from helical coiled tubes. Int J Thermal Sci, 43 (2004): 359–365.

[28] Rennie, T.J. and Raghavan, V.G.S. Experimental studies of a double pipe helical heat exchanger. Exp Thermal Fluid Sci, 29 (2005): 919–924.

[29] Rennie, T.J. and Raghavan, V.G.S. Numerical studies of a doublepipe helical heat exchanger. Appl Thermal Eng, 26 (2006a): 1266–1273.

[30] Rennie, T.J. and Raghavan, V.G.S. Effect of fluid thermal properties on heat transfer characteristics in a double pipe helical heat exchanger. Int J Thermal Sci, 45 (2006b): 1158– 1165.

[31] Rogers, G.F.C. and Mayhew, Y.R. Heat transfer and pressure drop in helically coiled tubes with turbulent flow. Int J Heat Mass Transf, 7 (1964): 1207–1216.

[32] Rustum, I.M. and Soliman, H.M. Numerical analysis of laminar mixed convection in horizontal internally finned tubes. Int J Heat Mass Transf, 33 (1990): 1485– 1496.

[33] Ruthven, D.M. The residence time distribution for ideal laminar flow in a helical tube. Chem Eng Sci, 26 (1971): 1113– 1121.

[34] Schmidt, E.F. Wfirmeiibergang und Druckverlust in Rohrschlangen. G’zemieJng. Tech., 39 (1967): 781– 789.

[35] Shah, L.J. Heat transfer correlations for vertical mantle heat exchangers. Solar Energy, 29 (2000): 157–171.

[36] Shah, R.K. and Joshi, S.D., , Convective heat transfer in curved ducts, in Handbook of single phase convective heat transfer, Kakac, S., Shah, R.K., & Hung, W. (eds). (Wiley Inter science, New York) (1987)

[37] Smith E.M., Advances in Thermal Design of Heat Exchangers. John Wiley & Sons Ltd. , 2005

[38] Van der Vyver, H., Dirker, J. and Myer, J.P. Validation of a CFD model for a three dimensional tube in tube heat exchanger, In Proc. 3rd International Conference on CFD in the Minerals and Process Industries, Australia, (2003) pp. 235–240.

[39] Xin, R.C., Awwad, A., Dong, Z.F. and Ebadian, M.A. An investigation and comparative study of the pressure drop in air water two phase flow in vertical helicoidal pipes. Int J Heat Mass Transf, 39(1996): 735–743.

[40] Yamamoto, K., Akita, T., Ikeuchi, H. and Kita, Y. Experimental study of the flow in a helical circular tube. Fluid Dyn Res, 16 (1995): 237–249.

[41] Yang, G., Dong, F., Ebadian, M.A. Laminar forced convection in a helicoidal pipe with finite pitch, Int. J. Heat Mass Transf. 38 (1995):853-862.