ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

CFD Analysis of Circular Tube with Different Strips and Nano Fluids

B Sravani
Department of Mechanical Engineering,
Sri Vaishnavi College of Engineering,
Singupuram, A.P - 532 185, India.

K. Srinivasa Rao
Department of Mechanical Engineering,
Sri Vaishnavi College of Engineering,
Singupuram, A.P - 532 185, India.

ABSTRACT

Heat exchangers are used in different process ranging from convection, utilization and recovery of thermal energy in various industrial, commercial and domestic applications. Some common examples include steam generation and condensation in power and cogeneration plants; sensible heating and cooling in thermal processing of chemical, pharmaceutical and agricultural products; fluid heating in manufacturing and waste heat recovery etc. The project was designed in CATIA V5 R20 and analysis was carried out by Fluent Software. By employing finite volume approach to solve the governing partial differential equation, heat transfer and fluid flow in laminar and turbulent regime were numerically carried out on induced tubes. Reynolds numbers between 1500 to 10000 were considered and the tubes were under uniform wall heat flux condition. As the Reynolds number increases both the Nusselt number and the friction factor decreases. The two types of tube design with (H/D=10& H/D=15) with three different Nano fluids. As result of improved swirl and superior temperature distribution with twisted tape (H/D=10) with Al_2O_3Nano fluid is the best performance. As the Reynolds number increases pressure drop decreases. From the CFD analysis observe that at Reynolds number 1500 and 0.8% CuO Nano fluid possess maximum pressure drop. It is compared with 1500 Reynolds number Al_2O_3, TiO_2 at (H/D=10) 11.1%, 33.5%. And also compared with (H/D=15) Al_2O_3, TiO_2, CuO is less than compared with CuO 52.3%, 29.15%, 7.5%.

Key words: ***heat exchangers, cfd, catia, pressure drop, twisted tape.***

1. INTRODUCTION

Fixed Geometry (FG) Criteria:

The area of flow cross-section (N and) and tube length L are kept constant. This criterion is typically applicable for retrofitting the smooth tubes of an existing exchanger with enhanced tubes, there by maintaining the same basic geometry and size (N, d, L). The objectives then could be to increase the heat load Q for the same approach temperature ΔT_i and mass flow rate m or pumping power P; or decrease ΔT_i or P for fixed Q and m or P; or reduce P for fixed Q.

1.2 TREATED SURFACES:

It consists of a variety of structured surfaces (continuous or discontinuous integral roughness or alterations) and coatings [1]. The roughness created by this treatment does not any significant effect in the single phase heat transfer. These are applicable in cases phase heat transfer only. If heat transfer coefficient is poor in annulus, axially finned inner tube (or tubes) can be used. Double-pipe heat exchangers are built in modular concept, i.e., in the form of hair fins.

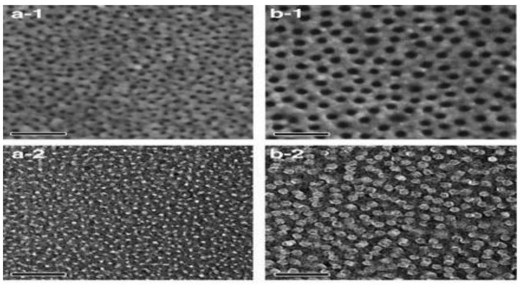

Fig 1.1: Corrugated tube Roughness

Cite this article as: B Sravani & K. Srinivasa Rao, "CFD Analysis of Circular Tube with Different Strips and Nano Fluids", International Journal & Magazine of Engineering, Technology, Management and Research, Volume 5 Issue 2, 2018, Page 85-94.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

EXTENDED SURFACES:

Extended or finned surfaces increase the heat transfer area which could be very effective in case of fluids with low heat transfer coefficients. This technique includes finned tube for shell & tube exchangers, plate fins for compact heat exchanger and finned heat sinks for electronic cooling [2].

Finned surfaces enhance heat transfer in natural or forced convection which can be used for cooling of electrical and electronic devices. The use of extended surfaces for cooling electronic devices is not restricted to the natural convection heat transfer regime but also can be used for forced convective heat transfer. Segmented or interrupted longitudinal fins, as shown in Fig2.2, promote boundary layer separation of the fluids and disturb the whole bulk flow field inside circular tubes. Separation and restarting of the boundary layers increases the heat transfer rate. Plate fin or tube and plate fin type of compact heat exchangers, where the finned surfaces provide a very large surface area density, are used increasingly in many automotive, waste heat recovery, refrigeration and air conditioning , cryogenic, propulsion system and other heat recuperative applications. A variety of finned surfaces typically used include offset strip fins, louvered fins, perforated fins and wavy fins [3].

Fig 1.2: Segmented fin heat sink

1.4 DISPLACED ENHANCEMENT DEVICES:

Displaced enhancement devices displace the fluid elements from the core of the channel to heated or cooled surfaces and vice versaDisplaced enhancement devices include inserts like static mixer elements (e.g.Kenics, Sulzer), metallic mesh, and discs, wire matrix inserts, rings or balls. Different types of conical ring inserts used in circular tubes are shown in Fig 2.3. These inserts do not alter heat transfer surface and provide a lot of scope for inter-mixing of the fluid particles. Disks promote higher heat transfer with moderate increase in friction factor whereas friction factor is very high for rings and round balls. Bergles found that pressure drop in the turbulent flows are extremely high. Most of the devices are suitable for laminar flow only. The main objective behind the use of static mixers is to increase the fluid mixing, so its application is limited to chemical processes with heat transfer only.

1.4 Log Mean Temperature Differences

Heat flows between the hot and cold streams due to the temperature difference across the tube acting as a driving force. As seen in the Figure 7.3, the temperature difference will vary along the length of the HX, and this must be taken into account in the analysis [4].

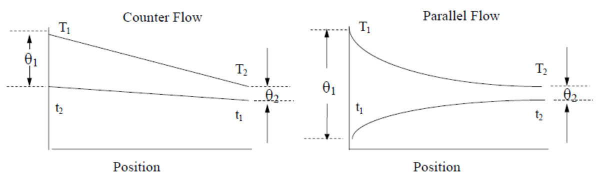

Fig.1.3 :Log Mean Temperature Differences

1.5: Temperature Differences between Hot and Cold Process Streams

From the heat exchanger equations shown earlier, it can be shown that the integrated average temperature difference for either parallel or counter flow may be written as:

$$\Delta\theta = LMTD = \frac{\theta_1 - \theta_2}{\ln\left(\frac{\theta_1}{\theta_2}\right)}$$

The effective temperature difference calculated from this equation is known as the log mean temperature

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

difference, frequently abbreviated as LMTD, based on the type of mathematical average that it describes. While the equation applies to either parallel or counter flow, it can be shown that Δθ will always be greater in the counter flow arrangement. Another interesting observation from the above Figure is that counter flow is more appropriate for maximum energy recovery [5].

2.LITERATURE REVIEW

Pak and cho et.al. [1] may be considered as a pioneering work in estimating the properties of Al_2O_3 nanofluid and determination of heat transfer coefficients in the turbulent range. A regression equation has been presented by the them for the determination of thermal conductivity valid for volume concentration Ø<3.0%

Yu and Choi et.al. [2] to predict the effective thermal conductivity of Nano fluids. They assumed the base fluid molecules close to the surface of the nanoparticles to form a solid like layered structure having thermal higher than that of the base fluid.

Koo and Kleinstreuer et.al. [3] explained the influence of Brownian motion, thermophoresis, osmo-phoresis and particle size on the thermal conductivity of Nano fluid. From their analysis the Brownian motion is found to be significant parameter by 10^6 and 10^8 orders more in comparison to thermo and osmo-phoresis respectively.

Gao and Zhou et.al. [4] considered both physical and geometrical anisotropy of the highly conducting nanoparticle inclusions, presented a theory and found that adjustment of nanoparticles shape is really helpful to achieve appreciable enhancement of effective thermal conductivity.

Ahmed Azari et al [5] They concluded that the experimental results of convective heat transfer coefficient and pressure drop of Nano fluids containing Al_2O_3,TiO_2 and Sio2 flowing through a straight pipe were experimentally investigated and the results showed that Al_2O_3 and TiO_2 de-ionized water increased heat transfer coefficient considerably, while the Sio2 Nano-fluids showed the opposite behavior, and also heat transfer coefficient is better than Tio2 Nano-fluid due to thermal conductivity of Al_2O_3.

Casquillas et al. [6]have been analyzed thermal conductivity of Nano fluid increases with increases with percentage of volume concentration.

Wang et al.[7]used Al_2O_3 And CuO nanoparticles in water, vacuum pump fluid, engine oil, and ethylene glycol for the estimation of thermal conductivity and observed that thermal conductivity of nanofluid is high compared to base fluid.

3.Catia Modeling

3.1MODELING:

In the process of the Catia modelling of pipe we have to design two Parts. They are,

Pipe

Twisted tape

Pipe length = 2000mm

Inner dia of pipe = 14.5mm

Outer dia = 16.5 mm

pipe thickness = 2mm

H/D ratio of tape = 10, 15

Tape thickness = 2mm

Used Catia Tools:

Plane, Project 3D Elements, Pad, Pocket and Rectangular Pattern.

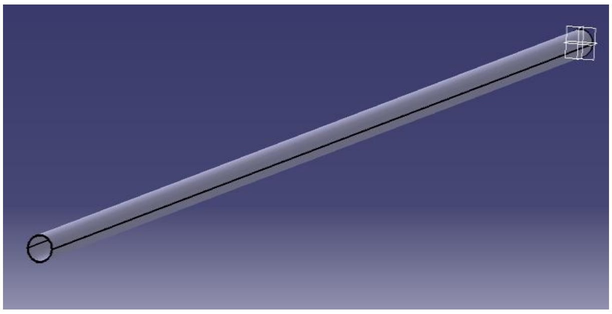

Fig 3.1 Designed Catia model of Pipe

Used Catia Tools:

Plane, Project 3D Elements, Pad, Pocket and Rectangular Pattern.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

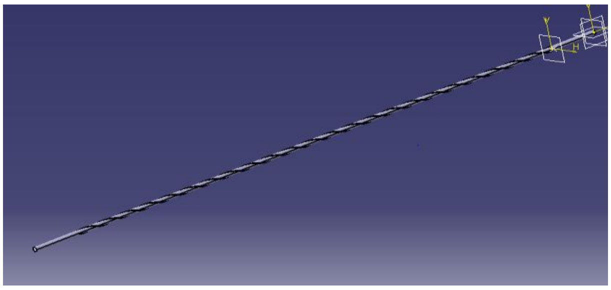

Fig 3.2 Designed Catia model strip

Used Catia Tools:
Plane, Project 3D Elements, Pad, Pocket and Rectangular Pattern.

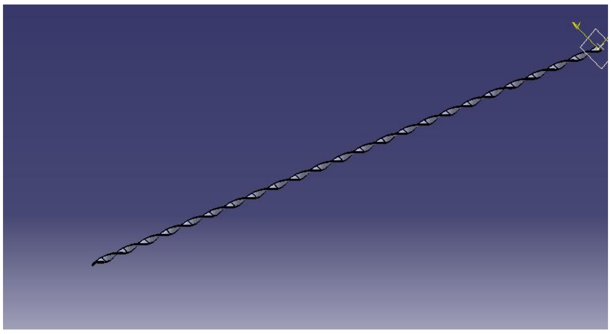

Fig 3.3 Designed Catia model of Twisted strip

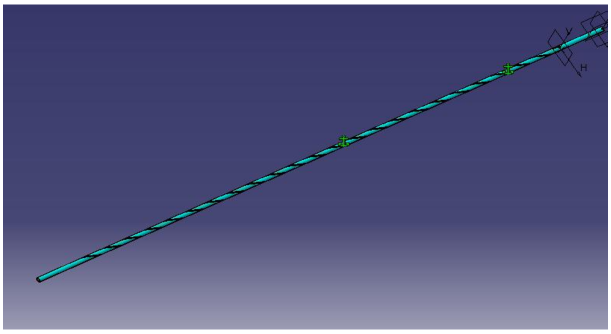

Fig 3.4 Designed Catia model of tube with strip

3.2 DESIGNED CATIA MODEL:

For this project, fully developed laminar and turbulent incompressible fluid flow will be analyzed in three heat exchanger cases: parallel flow, counter flow, and flow in a fouled heat exchanger. The resulting temperature difference will be compared and be determined as a function of the inlet velocity and inlet temperatures. The overall objective is to determine the max temperature difference in these cases for both laminar and turbulent flow for a variety of flow rates and inlet temperatures. To simplify the number of variables, water and oil will be chosen as the fluids to maintain viscosities and densities of the fluids constant. The type of heat exchanger used will be of concentric tube design. Water will be the cooling medium and oil the working fluid.

4. CFD ANALYSIS

Computational fluid dynamics (CFD) [6] is a computer-based simulation method for analysing fluid flow, heat transfer, and related phenomena such as chemical reactions. This project uses CFD for analysis of flow and heat transfer (not for analysis of chemical reactions). Some examples of application areas are: aerodynamic lift and drag (i.e. airplanes or windmill wings), power plant combustion, chemical processes, heating/ventilation, and even biomedical engineering (simulating blood flow through arteries and veins). CFD analyses carried out in the various industries are used in R&D and manufacture of aircraft, combustion engines, as well as many other industrial products.

This section briefly describes the general concepts and theory related to using CFD to analyse fluid flow and heat transfer, as relevant to this project. It begins with a review of the tools needed for carrying out the CFD analyses and the processes required.

4.1 GEOMETRY:

The tube with twisted tape is built in the ANSYS workbench design module [7]. First, the fluid flow (fluent) module from the workbench is selected. The design modeler opens as a new window as the geometry is double clicked.

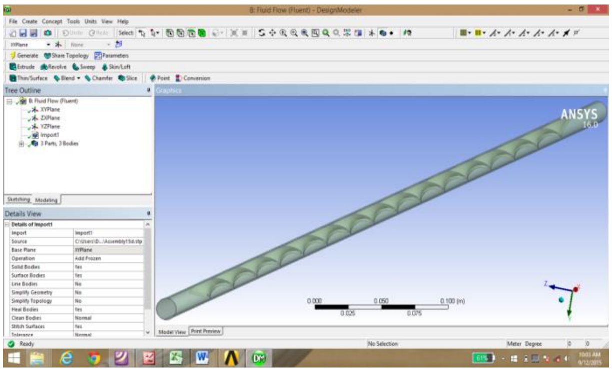

Fig. 4.1 Imported model in geometry

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

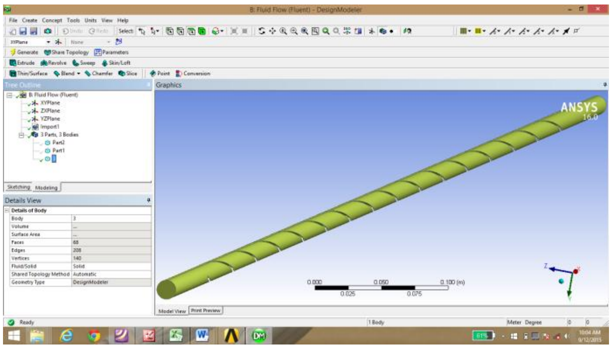

Fig. 4.2 fluid domain of pipe

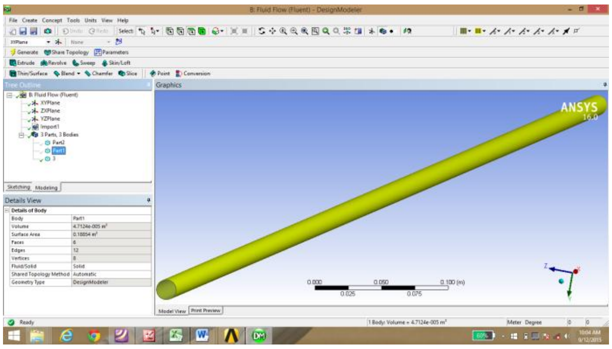

Fig. 4.3 imported fluid domain

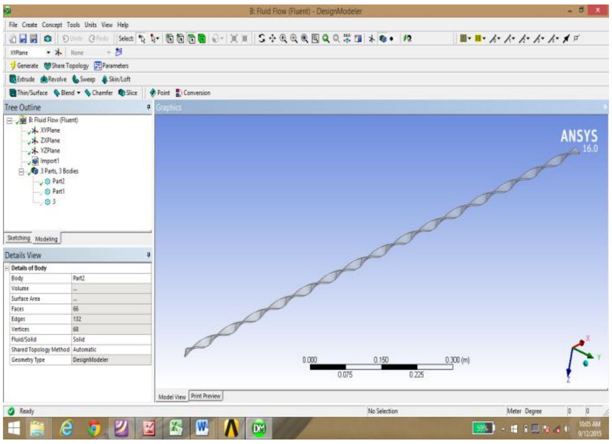

Fig. 4.4 imported model of strip

PART NUMBER	PART OF THE MODEL	STATE TYPE
1.	INNER FLUID	FLUID
2	OUTER TUBE	SOLID
3.	TWISTED TAPE	SOLID

Table.4.1 geometry type and model

4.1.1 The Main Solver

The solver is the heart of CFD software. It sets up the equation set according to the options chosen by the user and meshes points generated by the pre-processor, and solves them to compute the flow field. The process involves the following tasks:

- selecting appropriate physical model,
- defining material properties,
- prescribing boundary conditions,
- providing initial solutions,
- setting up solver controls,
- set up convergence criteria,
- solving equation set, and
- saving results

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

4.2 MESHING

Initially a relatively coarser mesh is generated. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured hexahedral cells as much as possible. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, a fine mesh is generated. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed [8].

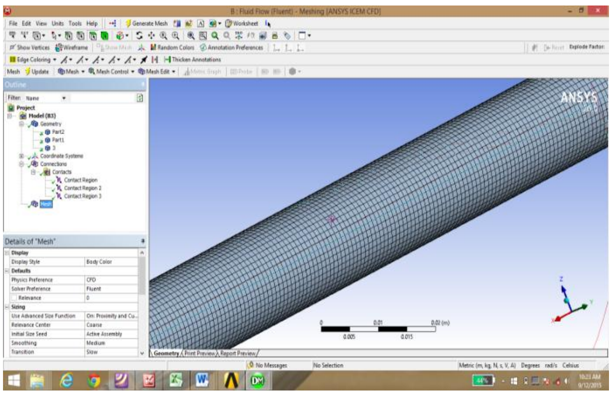

Fig 4.5 Pipe with twisted tape geometry full mesh

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

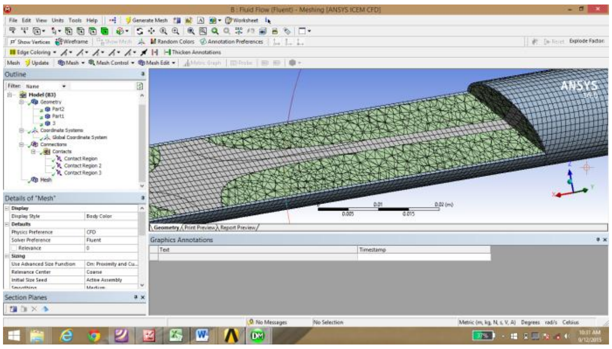

Fig 4.6 Cross section twisted tape mesh

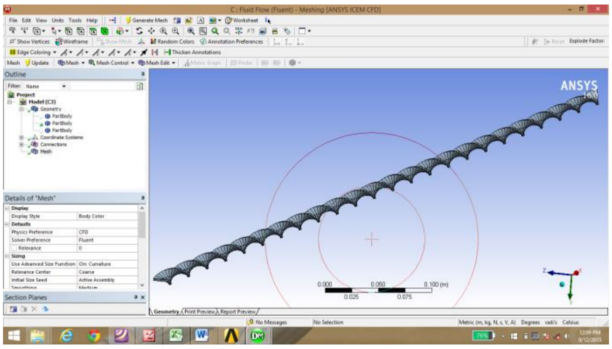

Fig 4.7 Twisted tape mesh

The different surfaces of the solid are named as per required inlets and outlets for inner and outer fluids.

Save project again at this point and close the window. Refresh and update project on the workbench. Now open the setup. The ANSYS Fluent Launcher will open in a window. Set dimension as 3D, option as Double Precision, processing as Serial type and hit OK. The Fluent window will open.

4.3 SETUP:

The mesh is checked and quality is obtained.

4.3.1 MATERIALS:

The create/edit option is clicked to add water-liquid, steel and copper to the list of fluid and solid respectively from the fluent database

4.3.2 CELL ZONE CONDITIONS:

In cell zone conditions, we have to assign the conditions of the liquid and solid.

Table 4.2 cell zone conditions

Sno.	PART/BODY	MATERIAL
1.	INNER FLUID	NANO FLUID
2.	OUTER TUBE	COPPER
3.	TWISTED TAPE	ALLUMINIUM

4.3.3 BOUNDARY CONDITIONS:

Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as velocity inlet and pressure outlet. As this is a counter-flow with two tubes so there are two inlets and two outlets [9]. The walls are separately specified with respective boundary conditions. No slip condition is considered for each wall. Except the tube walls each wall is set to zero heat flux condition. The details about all boundary conditions can be seen in the table as given below.

Table 4.3 boundary conditions

	BOUN DARY CONDITION TYPE	MASS FLOW RATE(kg/s)	TEMPERATURE (k)
IINLET	Mass flow inlet		303k
INNER OUTLET	Pressure outlet	-	-
OUTER INLET	Mass flow inlet	1	300
OUTER OUTLET	Pressure outlet	-	-

4.4 SOLUTION:

RUN CALCULATION:

After giving the boundary conditions to the inner and outer fluid, finally we have to run the calculations. The number of iteration is set to 500 and the solution is calculated and various contours, vectors and plots are obtained.

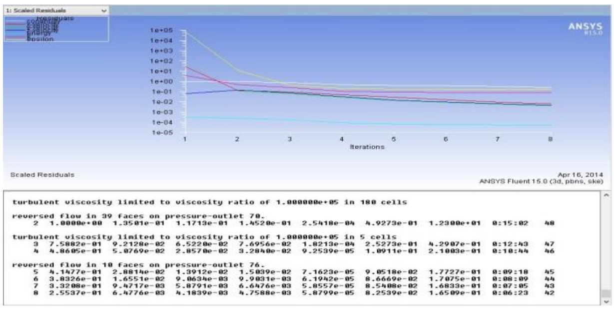

Fig 4.8 Calculations was running

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

4.5The Post-processor

The post-processor is the last part of CFD software. It helps the user to examine the results and extract useful data. The results may be displayed as vector plots of velocities, contour plots of scalar variables such as pressure and temperature, streamlines and animation in case of unsteady simulation. Global parameters like drag coefficient, lift coefficient, Nusselt number and friction 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.

Several general-purpose CFD packages have been published in the past decade. Prominent among them are:PHOENICS,FLUENT, STAR-CD,CFX, CFD-ACE,ANSWER, CFD++, FLOW-3D and COMPACT. Most of them 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 grid generators and post processors. Software such as ICEM CFD, Some popular visualization software used with CFD packages are TECPLOT and FIELDVIEW [10].

4.6 OVERVIEW OF FLUENT PACKAGE

FLUENT is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, solving your flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh 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. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all made possible. In addition, FLUENT uses a client/server architecture, which allows it to run as separate simultaneous processes on client desktop workstations and powerful compute servers, for efficient execution, interactive control, and complete flexibility of machine or operating system type. All functions required 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.

5. RESULTS AND DISCUSSIONS

FLUENT analysis of temperature difference in fluid flow the base conditions must first be analyzed. The first condition is that of fluid flowing through a tube with constant wall heat flux condition. For this project developing laminar and turbulent incompressible fluid flow was analyzed in tube fitted with two different twisted tape ratios. The overall objective of this project was to determine the heat transfer coefficient and friction factor in these cases for both laminar and turbulent flow for a three different water based Nano fluid concentrations and variety of Reynolds numbers.

To analyze the heat transfer in tube with different tape inserts by comparing the simulation result to the experimental calculations.

The schematic diagram of the experimental setup and the twisted tape is shown in Fig.5.1.1 (a). The setup consists of a chiller, collecting tank, water pump, flow meter, pressure transducer, control panel, and test section. A copper tube of 2 m length having ID=14.5mm and OD =16.5 mm enclosed with heaters and ceramic fiber insulation constitute the test section. The twisted tapes are manufactured with aluminum strips of 1 mm thick and 14.5 mm wide as shown in Fig .5.1.1(b). It was fabricated for different twist ratio 10 and 15 with the width, H/D=10, 15 respectively. The heat conduction in the aluminum tape along its length is assumed to be negligible.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

5.1 REPORTS:

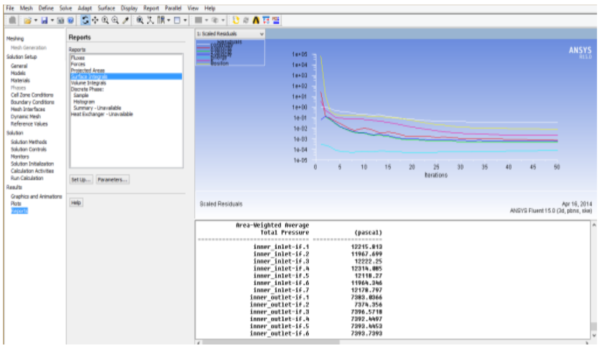

Fig 5.2 verifying the reports.

5.2 Full Pressure Contour

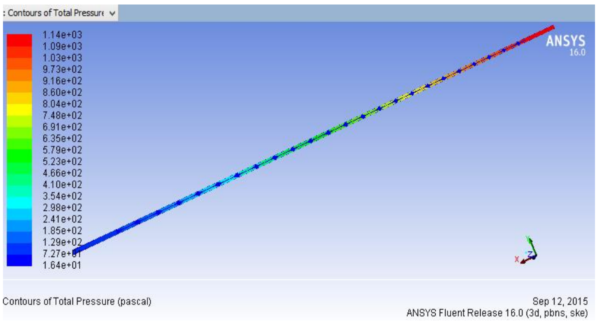

Fig 5.3 Full Pressure Contour

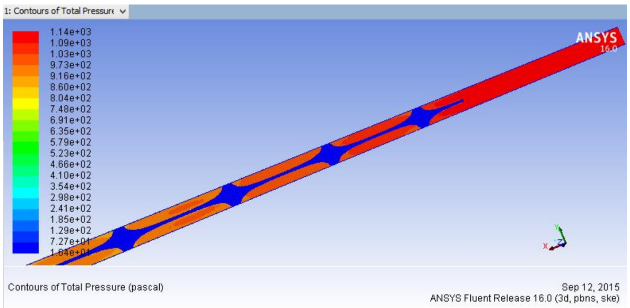

Fig 5.4 Pressure Contour starting

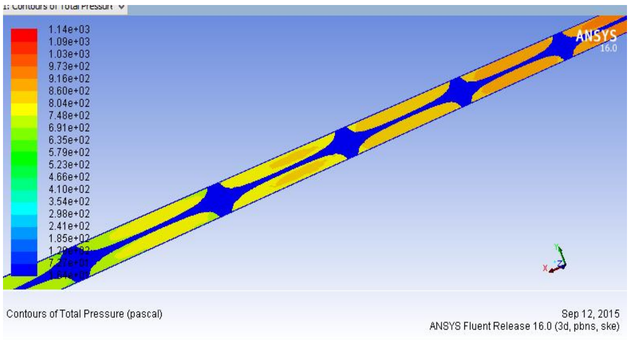

Fig 5.5 Full Pressure contour middle

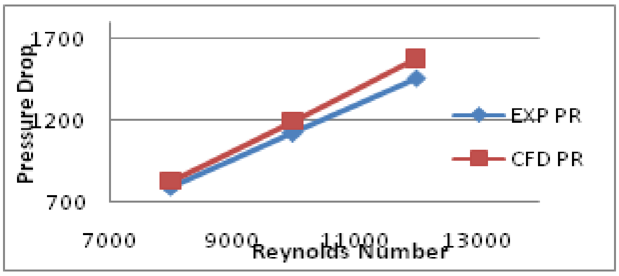

Fig 5.6 Variation of pressure drop with Reynolds number

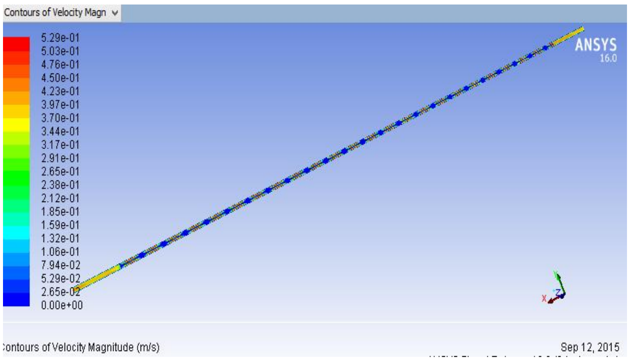

Fig 5.7 Full velocity contour

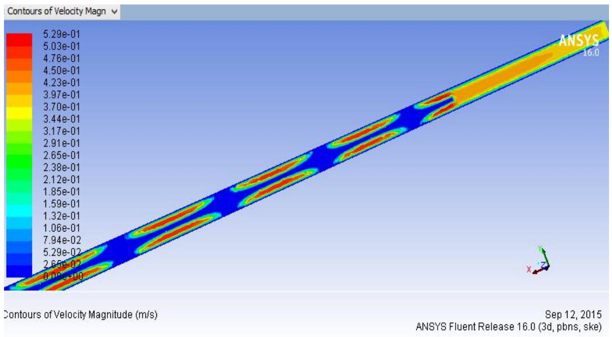

Fig 5.8 Velocity Contour Starting

5.3 Validation with Experimental Works

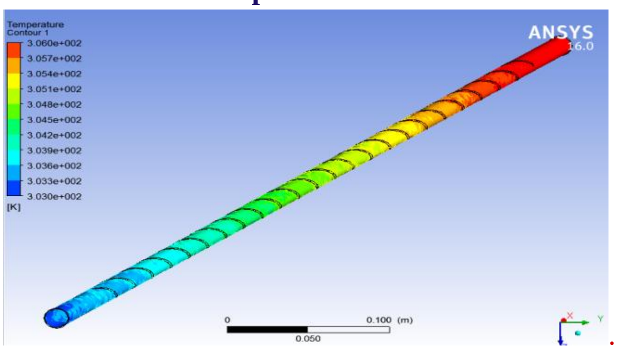

Fig 5.9 Temparature contour for the tube

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

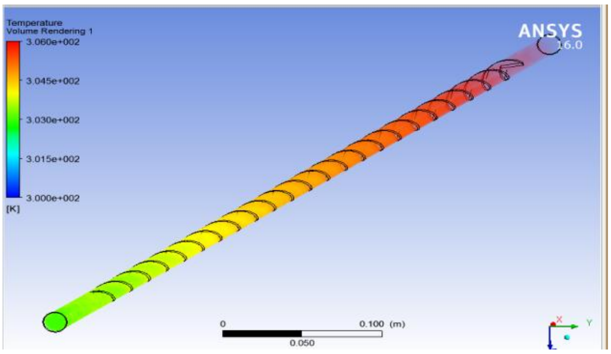

Fig 5.10 Temprature volume rendering

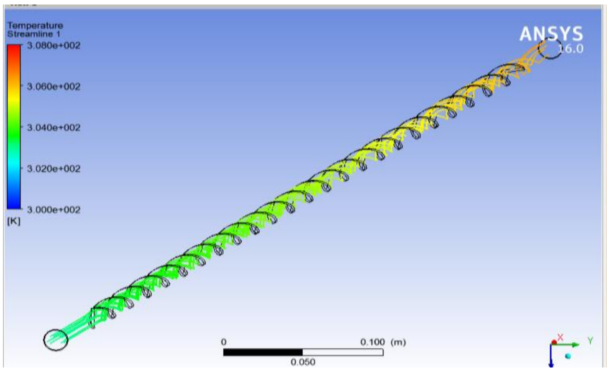

Fig 5.11 Temperature Streamline

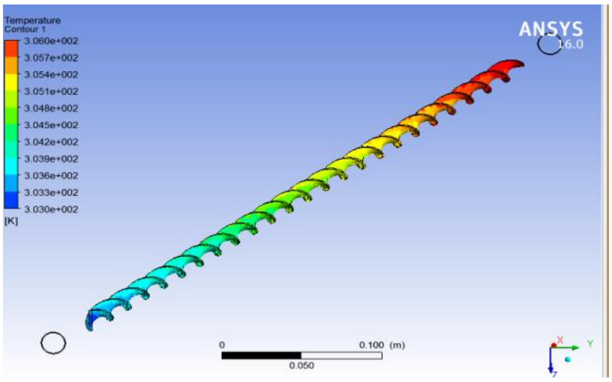

Fig 5.12 Velocity Streamline

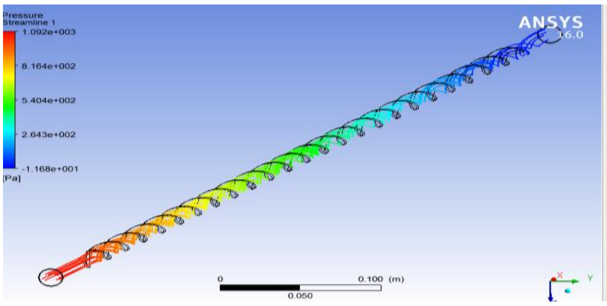

Fig 5.13 Temperature variation in pipe

With the aim of understanding the detailed flow physics and phenomena throughout the length of the domain the velocity vectors along with the magnitude of the stream wise velocity of the flow at a randomly selected axial location are presented in Figure 5.4.0 for all the pipe models undertaken. The randomly selected locations considered for each of the induced tube twisted ratios has either a different Nano fluids used .In the (H/D=10, 15) Figure 5.4.0, the velocity profile is almost the same at allocations and the maximum velocity occurs at the center implying that the flow is fully developed and parabolic at these locations. However, when the tube is induced with a plain twisted tape (Figure 5.4.0), the velocity near the wall at all the locations increases by about 6.17% of that in the (H/D=10, 15).

6.CONCLUSION

By employing finite volume approach to solve the governing partial differential equation, heat transfer and fluid flow in laminar and turbulent regime were numerically carried out on induced tubes. Reynolds numbers between 1500 to 10000 were considered and the tubes were under uniform wall heat flux condition. The turbulent variant of the K- ϵ model was selected to run the simulations. The Nusselt number, friction factor and pressure drop for laminar and turbulent regimes for different tube designs (H/D=10&15) were obtained. As the Reynolds number increases both the Nusselt number and the friction factor decreases. As the particle concentration increased heat transfer coefficient increases. In this best heat transfer coefficient is at 0.8c%Al_2O_3(H/D=10) Reynolds number 10000. These values compared with the CuO,TiO_2(H/D=10) and Al_2O_3, iO_2,CuO at (H/D=15) less compared to $Al_2O_3$10.95%,33.04%,4.2%,37.13%,12.83%. As the Reynolds number increases pressure drop decreases. From the CFD analysis observe that at Reynolds number 1500 and 0.8C%CuO Nano fluid possess maximum pressure drop. It is compared with 1500 Reynolds number Al_2O_3, TiO_2 at (H/D=10) 11.1%, 33.5%.And also compared with (H/D=15) Al_2O_3, TiO_2,CuO is less than compared with CuO52.3%,29.15%,7.5%.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

7. REFERENCES

[1] M Pak and cho, Performance Analysis Of Tube Heat Using Miscible System, American Journal Of Applied Sciences 5 (5): 548-552, 2008.

[2] Yu and Choi, Heat Transfer Optimization of Tube through CFD Studies, Chalmers University of Technology, 2011.

[3] Koo and Kleinstreuer, A Design And Rating Method of Tube With Helical strips, Journal Of Heat Transfer, May 2010.

[4] Muhammad Mahmoud AslamBhutta, Nasir Hayat, Muhammad Hassan Bashir, AhmerRais Khan, KanwarNaveed Ahmad, Sarfaraz Khan5, CFD Applications In Various Heat Exchangers Design: A Review, Department Of Mechanical Engineering, University Of Engineering & Technology, Applied Thermal Engineering, 2011.

[5] ŽarkoStevanović, GradimirIlić, Design of Shell-And-Tube Heat Exchangers by Using CFD Technique, University Of Niš, Fr, 2002.

[6] Ender Ozden, Ilker Tari, Shell Side CFD Analysis of a Small Shell and Tube Heat Exchanger, Middle East Technical University, 2010.

[7] Kern DQ. Process heat transfer. New York, N.Y.: McGraw-Hill; 1950.

[8] Emerson, W.H., “Shell-side pressure drop and heat transfer with turbulent flow in segmentally baffled shell-tube heat exchangers”, Int. J. Heat Mass Transfer 6 (1963), pp. 649–66.

[9] G. V. Gaddis ES, “Pressure drop on the shell side of shell-and-tube heat exchangers withsegmental baffles,” Chem Eng Process, vol. 36, pp. 149–59, 1997.

[10] S. Kakac and H. Liu, “Heat exchangers: Selection, rating, and thermal performance,” 1998.