EXPERIMENTAL ANALYSIS & SIMULATION OF DOUBLE PIPE HEAT EXCHANGER

Kale Shivam B [1], Kadam Prashant P [2], Pardeshi Rohansingh G [3], Karwande Swapnil C [4]

[1] *Student, Mechanical Engineering, GHRCOEM, Ahmednagar, Maharashtra, India*
[2] *Student, Mechanical Engineering, GHRCOEM, Ahmednagar, Maharashtra, India*
[3] *Student, Mechanical Engineering, GHRCOEM, Ahmednagar, Maharashtra, India*
[4] *Assistant Professor, Mechanical Engineering, GHRCOEM, Ahmednagar, Maharashtra, India*

ABSTRACT

Heat transfer simulation techniques refer to analyzing the heat exchangers overall performance by using simulation software's such as ANSYS, CFD Analysis, etc. Mainly these techniques are used for analyzing the various parameters of the heat exchanger and the comparison is done with the computational fluid dynamics and experimental results. The applications of heat exchanger are in Thermal Power Plants, Process Industries, Refrigerators, Air Conditioning Equipment, Automobiles, and Radars for Space Vehicles etc. In the last decade several studies on the on the CFD techniques of heat exchangers have successfully done by researchers. The present paper review mainly focuses on the heat transfer enhancement by checking it with ANSYS FLUENT 16.0 by using various types of flow and using twisted tape and without using twisted tape. By doing the CFD of the double pipe heat exchanger with various boundary conditions and various flow parameters the results are obtained and compared with computational fluid dynamics and experimental validation to know the actual variation of different parameters. Thus it would cause to select the appropriate heat exchanger; actual comparison can be done and compared. Actual parameters are found using the simulation techniques and the actual values are determined of the parameters such as Nusselt number, Effectiveness of the heat exchanger, overall heat transfer coefficient etc.

Keyword: - *Double Pipe Heat Exchanger, ANSYS FLUENT 16.0, CFD Analysis, Nusselt Number & Overall Heat Transfer Coefficient, etc.*

1. INTRODUCTION

A heat exchanger is a device that is used to transfer thermal energy (enthalpy) between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperatures and in thermal contact. Heat transfer between flowing fluids is one of the most important physical processes to study; various heat exchangers are used in many different types of industries such as the process industries, compact heat exchanger nuclear power plants, HVAC systems, food processing industries, etc. The main purpose of using a heat exchanger is to get an efficient method of transferring heat from one fluid to another, by direct or indirect contact. The heat transfer occurs by mainly three modes which are mainly: Conduction, Convection & Radiation. Conduction takes place when the heat flows from a high temperature fluid to a low temperature fluid through the surrounding solid walls. Conduction mainly occurs in solids. Convection is the mode of heat transfer in which the heat transfer takes place between the adjacent layers of fluid. Convection mainly occurs in fluids and plays a major role in the performance of heat exchanger. Radiation does not play a significant role in the heat transfer in heat exchanger and hence it is neglected.

The analysis done is mainly done on the heat exchanger depending on the type of flow of the fluid in the heat exchanger. The main types of heat exchanger depending upon the type of flow are as follows-

1. Parallel Flow Heat Exchanger: In a parallel flow as the name suggest, the two fluid streams (hot and cold) travel in the same direction. The two streams enter at one end and leave at the other end. The temperature difference between the hot and cold fluids goes on decreasing from inlet to outlet in this type of heat exchanger. Since this type of heat exchanger needs a large area of heat transfer, therefore, it is rarely used in practice.

2. Counter Flow Heat Exchanger: In a counter flow heat exchanger, the two fluids flow in opposite directions. The hot and cold fluids enter at the opposite ends. The temperature distribution between the two fluids remains more or less nearly constant. Hence, such heat exchangers are most favored for heating and cooling of fluids.

1.1 Introduction to CFD

Computational Fluid Dynamics (CFD) provides a qualitative and quantitative prediction of fluid flows by means of-

1. Mathematical Modeling (Partial Differential Equations)
2. Numerical Methods (Discretization and Solution Techniques)
3. Software Tools (Solvers, Pre & Post Processing Utilities)

Computational Fluid Dynamics is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations such as conservation of mass, conservation of momentum, conservation of energy, effects of body forces, etc. which governs these processes using numerical methods. ANSYS CFD solvers are based on the finite volume method. In this method domain is discretized into a finite set of control volumes. After those general conversion equations for mass, momentum, energy, species, etc. are solved on this set of control volumes.

1.2 Working Procedure of CFD Package

CFD packages are developed with numerical algorithms that have ability to tackle fluid flow problems. In order to provide easy access to their solving power, all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements: (a) a pre-processor, (b) a numerical algorithm or solver and (c) a post-processor.

2. CFD ANALYSIS

Computational Fluid Dynamics (CFD) is the Finite Element Analysis technique which is used in the study of the fluids and its flow. Generally finite element analysis means the analysis of any body by dividing the body into small parts known as elements. As the body has infinite particles it is impossible to study the action on the each particle hence to avoid this difficulty the body is divided into small elements to get the results. This process is known as the meshing. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modeling starts with describing the boundary and initial conditions for the dominion and leads to modeling of the entire system. Finally it is followed by the analysis of the results, conclusions and discussion.

2.1 Geometry

The tool used to build the geometry is ANSYS Workbench Design Modular. ANSYS Design Modeler is used for creating cad models & ANSYS Fluent 16.0 is used to calculate temperature drop across heat exchanger by performing CFD analysis. Outlet temperature for cold and hot pipes is measured & compared with the actual experimental data. Experimental Nu number and overall heat transfer coefficient also correlate with CFD results. The geometry with the actual size is draw. Geometry consists of the major three parts- large pipe, small pipe & twisted tape.

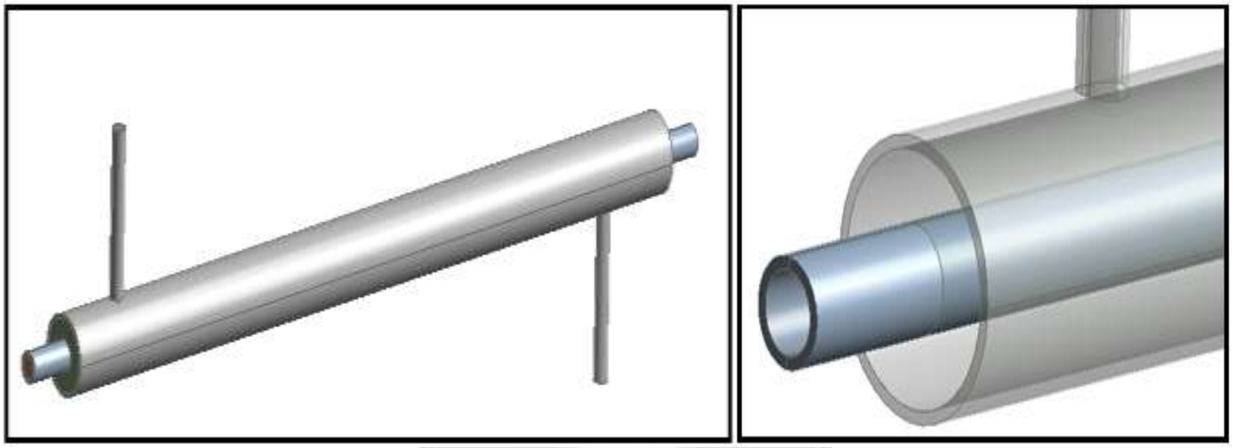

Fig -1 Geometry of the heat exchanger without twisted tape

Large Pipe & Small pipe geometry is drawn in Design Modular. As the geometry of the twisted tape is difficult to draw it is sketched in CATIA. The design is created by using wireframe model in CATIA. First, two helix are drawn of the given length and pitch at a distance equal to the width of the twisted tape. Then the sweep option is used to fill the geometry with the help of helix. The thickness is specified by using the offset command. The thickness of the twisted tape used is 2 mm. In this way the geometry of the double pipe heat exchanger model is created. The geometry of the twisted tape in CATIA is shown below.

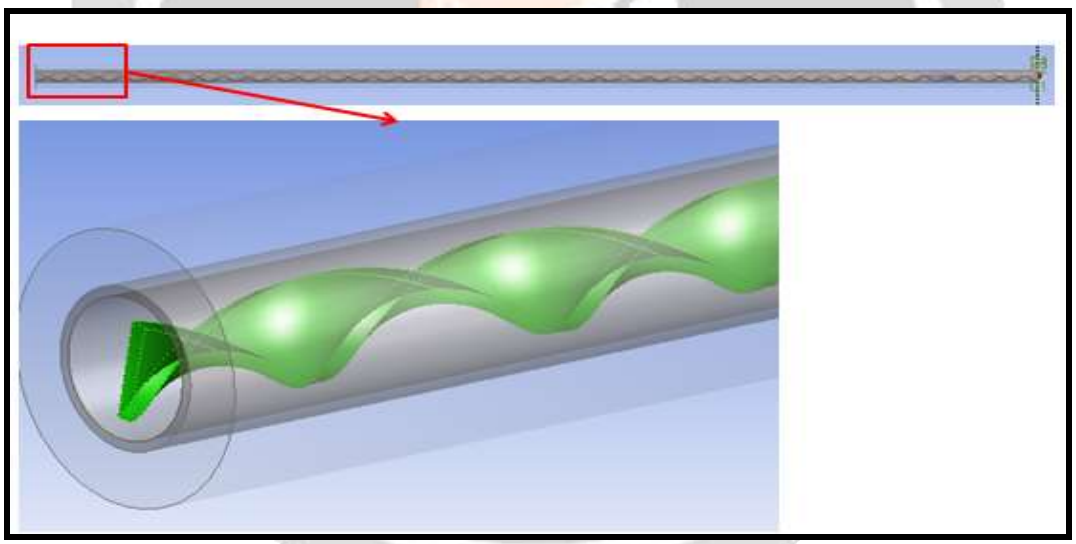

Fig -2 Close View of the twisted tape assembly in small pipe

2.2 Meshing

Meshing of the model consists of the discretization of the body into small parts known as elements. The meshing done for heat exchanger is volume meshing. As the fluid flows through the pipe the domain used for the analysis of the flow should be the volume in which the fluid is flowing. Hence the meshing done is of the parts which are in contact of the fluid or the volume in which the fluid is flowing. The tool used for meshing is ANSYS CFX which is a meshing software used for volume meshing. The meshing of the model without twisted tape and with twisted tape is done separately.

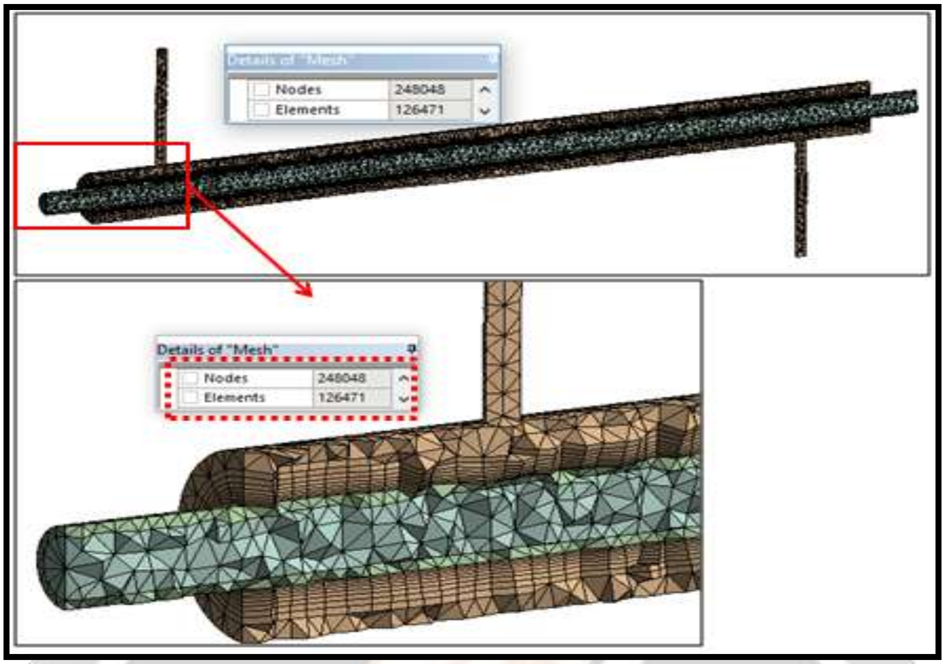

Fig -3 Meshing of plain heat exchanger

The geometry of the model of heat exchanger with twisted tape insert is complicated to mesh as the use of twisted tape makes it difficult to mesh. As the twisted tape geometry is not like a plain surface the meshing done is fine in this region. The number of nodes and elements produced in this mesh is much higher than that of the plain model due to the use of twisted tape. The figure of the mesh of the double pipe heat exchanger with twisted tape is shown below.

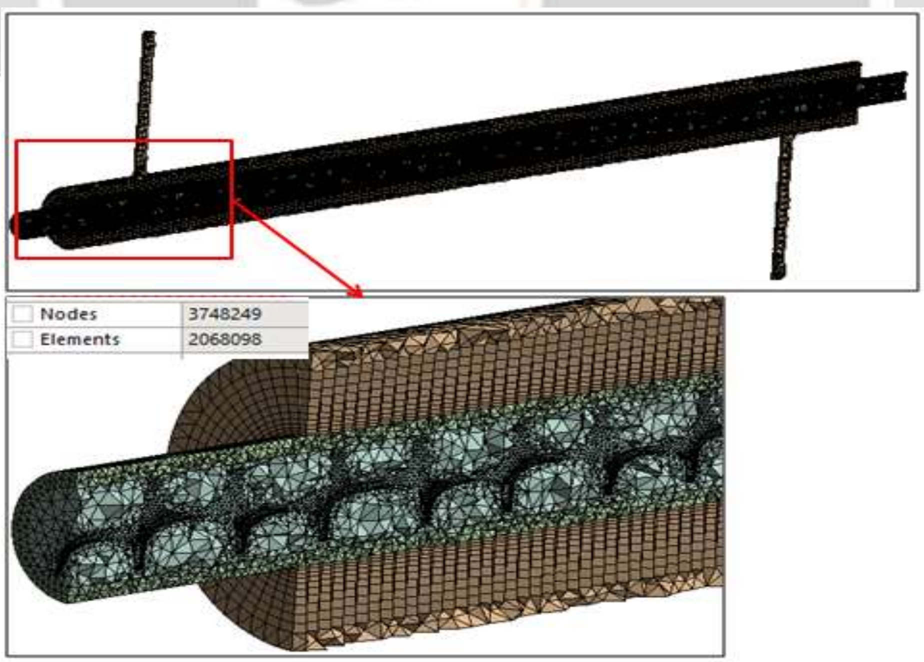

Fig -4 Mesh of heat exchanger with twisted tape

3. RESULTS AND DISCUSSON

The analysis is done for various mass flow rates for parallel and counter flow respectively with and without using twisted tape. The plots for the various outputs are plotted. It includes the temperature plot, Nusselt number plot & the overall heat transfer coefficient plot. The contours are plotted for each results obtained. The experimental input data was given as the input to the software as discussed earlier. The contours of different readings are discussed in the following section.

3.1 Experimental Results

The output results of the experimental performance is calculated by mathematical calculations and the values of the parameters such as effectiveness, Nusselt number, overall heat transfer coefficient, and the output temperature are shown in the table below.

CASES	MASS FLOW RATE (Kg/s)	T_{hi} (°C)	T_{ho} (°C)	T_{ci} (°C)	T_{co} (°C)	ε (%)	U_i (W/m^2°C)	Nu EXP
Counter Flow Without Twisted Tape	0.05	73	65	30	37	18.60	508.59	19.51
	0.10	70	62	30	37	20	1111.08	42.79
	0.15	70	59	30	36	27.50	1952.75	75.22
Counter Flow with Twisted Tape	0.05	76	64	30	40	26.09	756.74	28.98
	0.10	73	60	30	38	30.23	1558.47	59.90
	0.15	70	56	30	39	35	2921.43	112.65

Table -1 Experimental result for counter flow with & without twisted tape

CASES	MASS FLOW RATE (Kg/s)	Thi (°C)	Tho (°C)	Tci (°C)	Tco (°C)	ε (%)	Ui (W/m2°C)	Nu EXP
Parallel Flow Without Twisted Tape	0.05	72	63	30	35	21.43	481.96	18.33
	0.10	70	61	30	36	22.50	1111.78	42.31
	0.15	70	61	30	38	22.50	1948.81	74.65
Parallel Flow with Twisted Tape	0.05	74	65	30	37	20.45	535.06	20.34
	0.10	74	64	30	38	22.73	1238.30	47.14
	0.15	73	62	30	39	25.58	2188.99	83.84

Table -2 Experimental results for parallel flow with & without twisted tape

3.2 CFD Results

The CFD results are plotted by considering the input temperatures of the experimental analysis. The output temperatures are plotted by the software and also the parameters such as Nusselt number, Effectiveness & overall heat transfer coefficient are plotted and validated through the experimental results. The temperature contours and the Nusselt number contours are shown for the parallel and counter flow for plain and with twisted tape readings. The plots for the counter flow arrangement is shown only as it is inconvenient to show all the plots for 12 different results.

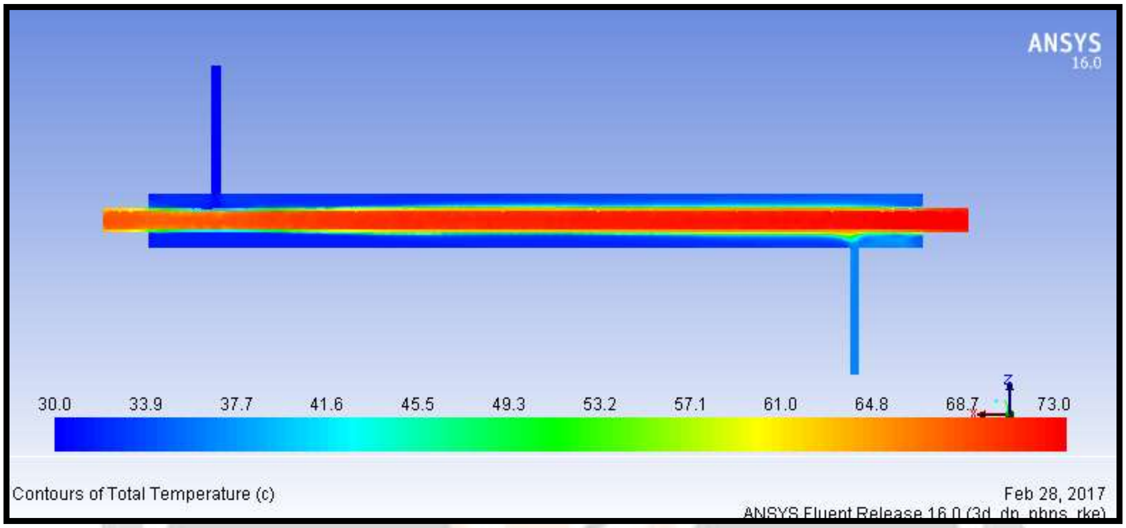

Fig -5 Temperature plot for counter flow without twisted tape

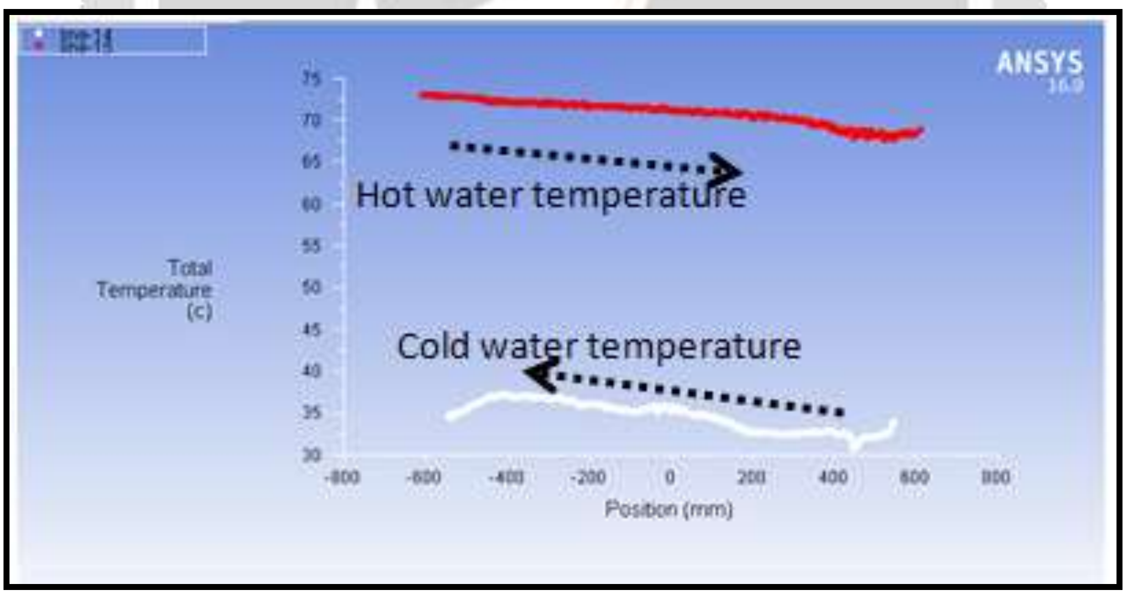

Fig -6 Temperature profile for counter flow without twisted tape

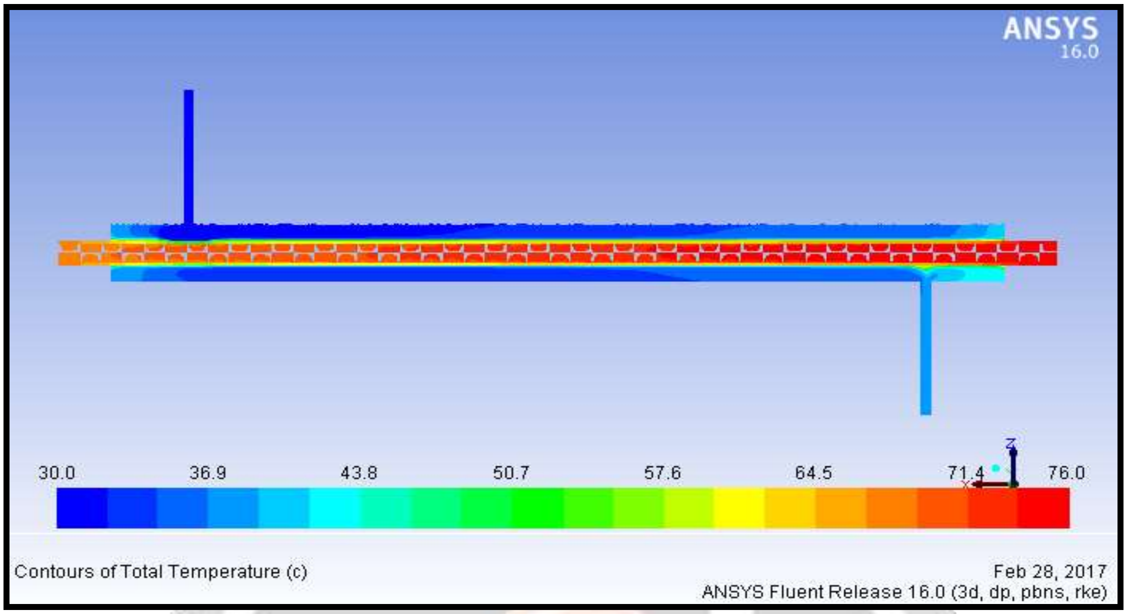

Fig -7 Temperature plot for counter flow with twisted tape

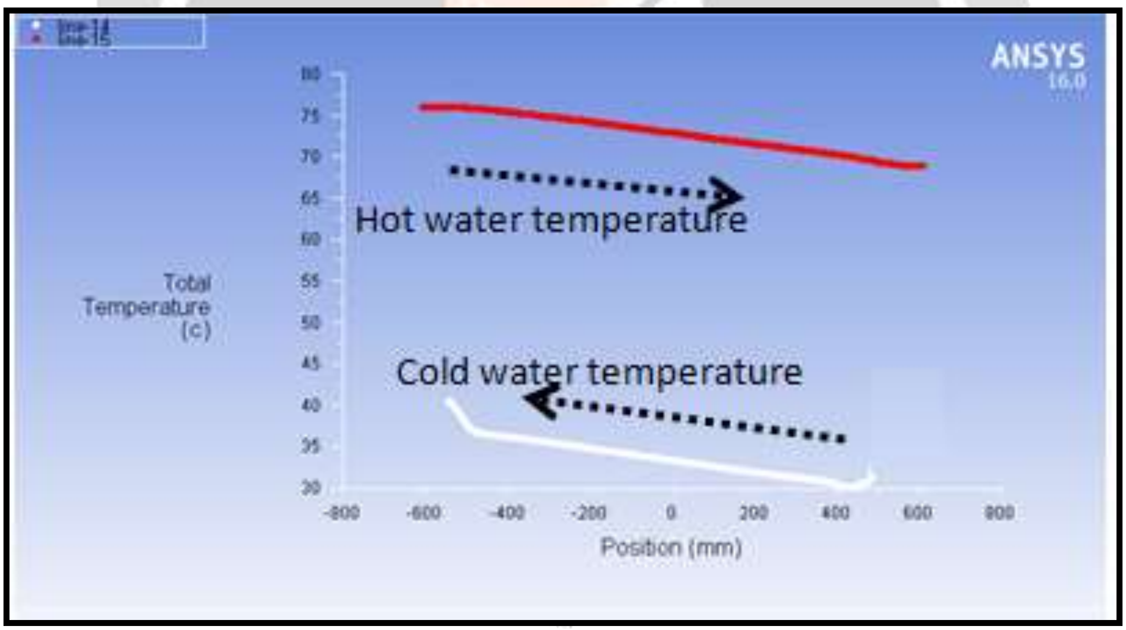

Fig -8 Temperature profile for counter flow with twisted tape

The output of the analysis done by the CFD is tabulated below for the counter & parallel flow with & without twisted tape. This mainly includes the temperature input and output, effectiveness, Nusselt number & overall heat transfer coefficient. The output results are shown below.

CASES	MASS FLOW RATE (Kg/s)	T_{hi} (°C)	T_{ho} (°C)	T_{ci} (°C)	T_{co} (°C)	ε (%)	U_i (W/m^2°C)	Nu CFD
Counter Flow Without Twisted Tape	0.05	73	68.7	30	36.8	16.35	469.34	20.40
	0.10	70	64.7	30	36.2	17.54	1021.84	44.60
	0.15	70	54.8	30	34.7	19.81	1779.24	78.80
Counter Flow with Twisted Tape	0.05	76	68.6	30	37.5	21.45	655.71	30.50
	0.10	73	66	30	35.7	21.89	1345.45	61.40
	0.15	70	61.9	30	38.5	26.37	2578.83	114.50

Table -3 CFD results for counter flow with & without twisted tape

CASES	MASS FLOW RATE (Kg/s)	T_{hi} (°C)	T_{ho} (°C)	T_{ci} (°C)	T_{co} (°C)	ε (%)	U_i (W/m^2°C)	Nu CFD
Parallel Flow Without Twisted Tape	0.05	72	67.7	30	34.3	14.86	423.43	19.60
	0.10	70	65.7	30	34.9	16.93	992.61	45.30
	0.15	70	65.5	30	37.9	19.66	1799.36	80.40
Parallel Flow with Twisted Tape	0.05	74	66.7	30	37.4	17.06	500.85	21.10
	0.10	74	67.3	30	38.4	19.25	1167.25	48.90
	0.15	73	65.5	30	39.7	22.40	2106.60	85.70

Table -4 CFD results for parallel flow with & without twisted tape

4. VALIDATION GRAPHS

The heat transfer validation is generally done by the changes in the values of dimensionless numbers. The dimensionless numbers such as Nusselt number, Reynolds number, etc are used to validate the output. The terms such as overall heat transfer coefficient and effectiveness gives the performance output of the heat exchanger hence they are compared for validation. The validation of the results is done by comparing it with the experimental results.

4.1 Comparison of the CFD Results with Experimental Results:

The comparison of the results of the experimental output & the output of the CFD is done for the heat transfer coefficient, Nusselt number & effectiveness. The change in the values is within ±15% which is acceptable due to the reasons explained below. The changes are shown in the table below.

CASES	MASS FLOW RATE (Kg/s)	Nu EXP	Nu CFD	% CHANGE (%)
Counter Flow Plain	0.05	19.51	20.40	4.36
	0.10	42.79	44.60	4.05
	0.15	75.22	78.80	4.54
Parallel Flow Plain	0.05	18.33	19.60	6.47
	0.10	42.31	45.30	6.60
	0.15	74.65	80.40	7.15
Counter Flow Twisted Tape	0.05	28.98	30.50	4.98
	0.10	59.90	61.40	2.44
	0.15	112.65	114.50	1.61
Parallel Flow Twisted Tape	0.05	20.34	21.10	3.60
	0.10	47.14	48.90	3.59
	0.15	83.84	85.70	2.17

Table -5 Nusselt number comparison

It can be seen from the above table that the Nusselt number found by the CFD is nearby the value of the experimental value & is within the limit. Hence we can say that the experimental setup is within the range of 1 to 7 % which is not varying much from the experimental results. The reasons for such variation are the losses occurring due to the atmospheric conditions or due to improper interpretation of the readings or due to human errors.

CASES	MASS FLOW RATE (Kg/s)	Ui EXP ($W/m^2 °C$)	Ui CFD ($W/m^2 °C$)	% CHANGE (%)
Counter Flow Plain	0.05	508.59	469.34	7.71
	0.10	1111.08	1021.84	8.03
	0.15	1952.75	1779.24	8.88
Parallel Flow Plain	0.05	481.96	423.43	12.14
	0.10	1111.78	992.61	10.71
	0.15	1948.81	1799.36	7.66
Counter Flow Twisted Tape	0.05	756.74	655.71	13.35
	0.10	1558.47	1345.45	13.66
	0.15	2921.43	2578.83	11.72

Parallel Flow Twisted Tape	0.05	535.06	500.85	6.39
	0.10	1238.30	1167.25	5.73
	0.15	2188.99	2106.60	3.76

Table -5 Overall heat transfer coefficient comparison

It can be seen from the above table that the overall heat transfer coefficient found by the CFD is nearby the value of the experimental value & is within the limit. Hence we can say that the experimental setup is within the range of 3 to 13 % which is not varying much from the experimental results.

4.2 Validation Graph

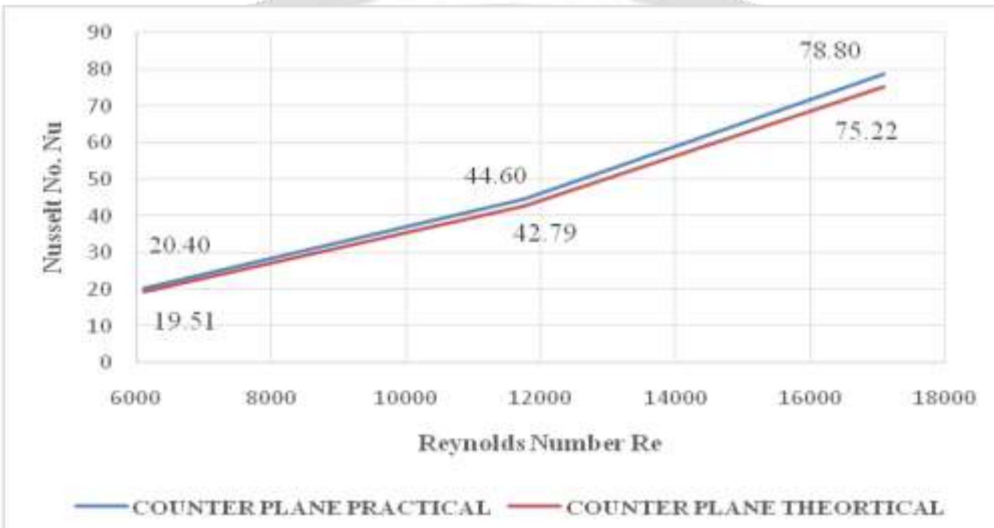

Chart -1 Nu Vs Reynolds Number for Counter flow Plane tube

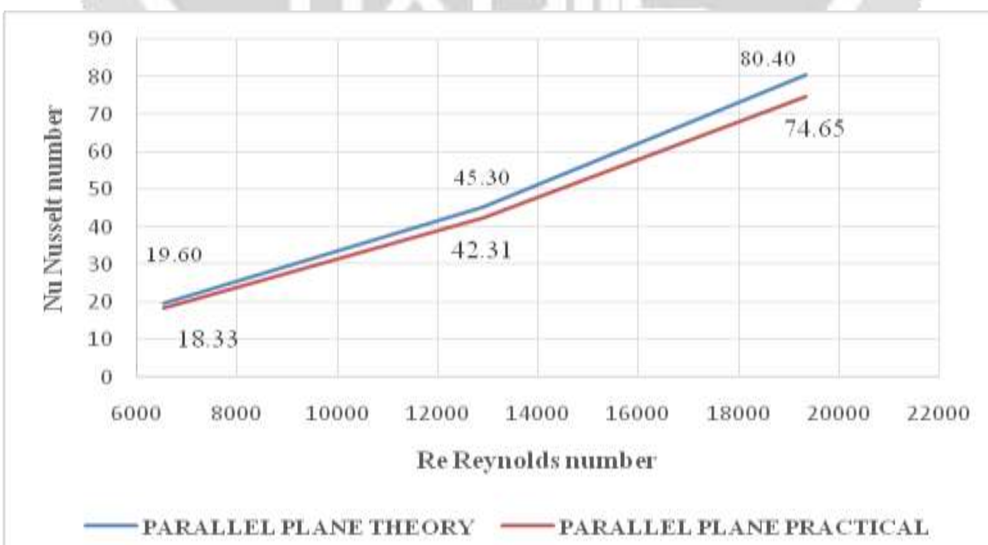

Chart -2 Nu Vs Reynolds Number for Parallel flow Plane tube

From the Chart -1 it has been observed that the theoretical values of Nusselt number for the given Reynolds number range from 6000 to 19000 is within 4.23% to 4.76% & from the Chart -2 graph it has been observed that the theoretical values of Nusselt number for the given Reynolds number range of 6000 to 19000 is within 6.91 to 7.70%.

5. CONCLUSIONS

After the CFD analysis and the validation with the experimental results following conclusions were obtained. The turbulence was created due to the insertion of twisted tape which in turn increased the heat transfer rate & hence increased the effectiveness of the double pipe heat exchanger.

1. The CFD results when compared with the experimental results for different conditions were found to be within the error limits.
2. The Nusselt number variation between the experimental and CFD results were within 1 to 7 %.
3. The variation of overall heat transfer coefficient was within 3 to 13 % with the experimental results which was found satisfactory.
4. Effectiveness of the heat exchanger calculated by the experimental and CFD were within 5 to 11 %.
5. The temperatures obtained from the CFD were within the limits and was accepted. The variation was not much than the experimental readings.
6. The accurate results were obtained by the use of CFD technique instead of theoretical calculations.

6. REFERENCES

[1]. Melvinraj C R, Vishal Varghese C & Vicky Wilson; *Comparative Study of Heat Exchangers Using CFD*; Int. Journal of Engineering Research and Applications; ISSN: 2248-9622, Vol. 4, Issue 5(Version 4), May 2014, pp.118-120.

[2]. Jibin Johnson, Abdul Anzar V M, Abith Shani, Harif Rahiman P; *CFD Analysis of Double Pipe Heat Exchanger;* International Journal of Science, Engineering and Technology Research (IJSETR), Volume 4, Issue 5, May 2015 1283.

[3]. M.Z.M.Saqheeb Ali, Mohan Krishna, D.V.V.S.Bhimesh Reddy; *Thermal Analysis of Double Pipe Heat Exchanger by Changing the Materials Using CFD;* International Journal of Engineering Trends and Technology (IJETT) – Volume 26 Number 2- August 2015.

[4]. Nice Thomachan, Anoop.K.S, Deepak.C.S; *CFD Analysis Of Tube In Tube Heat Exchanger With Fins;* International Research Journal of Engineering and Technology (IRJET); e-ISSN: 2395 -0056 Volume: 03 Issue: 04 Apr-2016.

[5]. J.S. Jayakumar, S.M. Mahajani, J.C. Mandal; *Experimental and CFD estimation of heat transfer in helically coiled heat exchangers;* chemical engineering research and design 8 6 (2008) 221–232.

[6]. Agniprobho Mazumder, Dr. Bijan Kumar Mandal; *Numerical Modeling and Simulation of a Double Tube Heat Exchanger Adopting a Black Box Approach;* Int. Journal of Engineering Research and Applications; ISSN: 2248-9622, Vol. 6, Issue 4, (Part - 2) April 2016, pp.35-41.

[7]. Mayank Bhola, Mr. Vinod Kumar and Dr. Satyendra Singh; *Heat Transfer Enhancement in Concentric Tube Heat Exchanger in ANSYS FLUENT 14.5;* IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 2 Issue 3, March 2015.

[8]. Sadashiv, Madhukeshwara.N; *Numerical Simulation of Enhancement of Heat Transfer in a Tube with And without Rod Helical tape Swirl Generators;* International Journal of Research in Aeronautical and Mechanical Engineering; Vol.2 Issue.1, January 2014; Pgs: 112-128.