A COMPARISION OF SHELL AND TUBE TYPE HEAT EXCHANGER BY EXPERIMENTAL AND CFD ANALYSIS

[1]PUSHPARAJ SINGH, [2]MUKESH KUMAR GAUTAM

[1]ASSISTANT PROFESSOR, [2]M.TECH. REASEACHER
REWA INSTITUTE OF TECH. REWA (MP)

***Abstract*: In these researches work a shell and tube type single pass heat exchanger considered for the comparative analysis. Analysis has been perform in two different phases, in first phase we prepare one setup of shell and tube type heat exchanger with brass tube for shell for cold water and straight copper tube of 500 mm for hot water due to its good thermal conductivity. Here water has taken as fluid media at different temperature. In order to perform the experiment we had prepare 12 different combination of velocity at inlet of hot and cold fluid along with the different inlet temperature. After the experimental study a computational fluid dynamic analysis was perform by creating a virtual model in CFD environment. The CFD model has created according to the physical parameter of experimental setup and same boundary condition has provided to analysis the performance of heat exchanger. The solution obtained for each combination of velocity and temperature input and corresponding output is stored in the form of solution table and graph. After the CFD analysis a comparative study has been performed to know the effectiveness of heat exchanger.**

***Keywords*: CFD, HEAT EXCHANGER, HEAT TRANSFER COEFFICIENT, LMTD, HOTWATER, COLD WATER.**

1. INTRODUCTION

Heat Exchanger:- A heat exchanger is a device built for efficient heat transfer from one medium to another medium. The media may be separated by a solid wall, so that they never mix, or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning, power plants, chemical plants, petrochemical plants, petroleum refineries, natural gas processing, cryogenics applications and sewage treatment. One common example of a heat exchanger is the radiator in a car, in which the heat source, being a hot engine-cooling fluid, water, transfers heat to air flowing around the radiator (i.e. the heat transfer medium).
The heat exchanger accepts two or more streams, which may flow in directions parallel or perpendicular to one another. When the flow directions are parallel, the streams may flow in the same or in opposite sense. Thus we can think of three primary flow arrangements:

a) Parallel flow
b) Counter flow
c) Cross flow

Thermodynamically, the counter flow arrangement provides the highest heat (or cold) recovery, while the parallel flow geometry gives the lowest. The cross flow arrangement, while giving intermediate thermodynamic performance, offers superior heat transfer properties and easier mechanical layout. Under certain circumstances, a hybrid cross counter flow geometry provides greater heat (or cold) recovery with superior heat transfer performance

2. LITERATURE REVIEW

Nitesh B. Dahare and Dr. M. Basavaraj, [1], This paper presents a recent innovation and development of a new technology, known as Twisted Tube technology, which has been able to overcome the limitations of the conventional technology, and in addition, provide superior overall heat transfer coefficients through tube side enhancement. This paper compares the construction, performance, and economics of Twisted Tube exchangers against conventional designs for copper materials of construction including reactive metals.

Durgesh Bhatt and Priyanka M Javhar [2], This paper presents the design of a shell-and- tube heat exchanger usually involves a trial and error procedure where for a certain combination of the design variables the heat transfer area is calculated and then another combination is tried to check if there is any possibility of increasing the heat transfer coefficient. Since several discrete combinations of the design configurations are possible, the designer needs an efficient strategy to quickly locate the design configuration having the minimum heat exchanger cost. In this particular problem the tube metallurgy and baffle spacing are being changed the results are obtained. In current paper the baffle spacing and tube metallurgy are the parameters considering change and effect of the same of heat transfer coefficient have been considered.

Thavamani J, at all, [3], In this paper the shell and tube heat exchanger (STHE) has been designed to cool the water from 60^{o} C to 51^{o} C. The experimental work is fabricated with the components of the exact dimensions as derived from the designing using CATIA and ANSYS. Tests are conducted on heat exchanger under the direction of flow, insulations under the atmosphere conditions. The observations and the result are discussed in the paper.

Guihong Pei and Liyin Zhang [4], the influence on the rock-soil temperature is approximately 13 % higher for the double-U heat exchanger than that of the single-U heat exchanger. The extracted energy of the intermittent operation is 36.44 kw·h higher than that of the continuous mode, although the running time is lower than that of continuous mode, over the course of 7 days. The thermal interference loss and quantity of heat exchanged for unit well depths at steady-state condition of 2.5 De, 3 De, 4 De, 4.5 De, 5 De, 5.5 De and 6 De of side tube spacing are detailed in this work. The simulation results of seven working conditions are compared. It is recommended that the side-tube spacing of double-U underground pipes shall be greater than or equal to five times of outer diameter (borehole diameter: 180 mm).

Muhammad Mahmood Aslam Bhutta [5], This literature review focuses on the applications of Computational Fluid Dynamics (CFD) in the field of heat exchangers. It has been found that CFD has been employed for the following areas of study in various types of heat exchangers: fluid flow distribution, fouling, pressure drop and thermal analysis in the design and optimization phase. Different turbulence models available in general purpose commercial CFD tools i.e. standard, realizable and RNG k _ ε RSM, and SST k _ ε in conjunction with velocity-pressure coupling schemes such as SIMPLE, SIMPLEC, PISO and etc. have been adopted to carry out the simulations. The quality of the solutions obtained from these simulations are largely within the acceptable range proving that CFD is an effective tool for predicting the behavior and performance of a wide variety of heat exchangers.

Umang K Patel [6], In this work, Heat exchangers are the important engineering systems with wide variety of applications including power plants, nuclear reactors, refrigeration and air- conditioning systems, heat recovery systems, chemical processing and food industries. Helical coil configuration is very effective for heat exchangers and chemical reactors because they can accommodate a large heat transfer area in a small space, with high heat transfer coefficients. Working towards the goal of saving energies and to make concise design for mechanical and chemical devices and plants, heat transfer play major role in design of heat exchangers. We are not use application of external power, but we can improve the heat transfer rate by modifying the design by providing the helical tubes, extended surface or swirl flow devices. We improve the heat transfer rate from helical coil tube-in-tube heat exchangers to use Computational Fluid Dynamics (CFD). My project aims to perform a numerical study of helical coil tube-in-tube heat exchanger with water as both hot and cold fluid. To improve the effectiveness, heat transfer rate and reduce power consumption, D/d geometrical parameter will be varied for different boundary conditions. The impact of this modification on Nusselt number, friction factor, and pumping power required and LMTD variation of inner fluid with respect to Reynolds number was studied.

3. PROBLEM IDENTIFICATION

Heat exchanger is one of the most important equipment in steel, power and many other industries due to its applications. Many researchers had worked on this topic and several research is also going on currently. Here we also try to improve the efficiency and effectiveness of heat exchanger by applying experimental and numerical technique. An experimental setup is installed for this research work and in order to find out the effectiveness of heat exchanger, we perform experiment by using different flow criteria and also by changing the velocity in each step.. After the experimental result we compare the result with the Numerical solution that we get from the CFD. We found very close relationship between these two results.

4. OBJECTIVE

Effectiveness of heat exchanger can be increased without significant change in the circuit is possible by only one way i.e. by increasing flow of cold fluid in the inlet valve. However if we increase the flow in cold inlet in the tube type Heat Exchanger, then characteristics goes down which could be balanced by decreasing no. of tube of Heat Exchanger and decreasing the relative velocity of fluids flowing in Tube Type Heat Exchange.

5. EXPERIMENTAL SET UP

The pipe used to construct the tube has 10 mm I.D. and 12.7mm O.D. The tube material is SS 316. The horizontal of the tube is 500mm and total 24 tubes used to create the required equipment of HE. The remaining parts of the setup are made of SS 304.The horizontal tube is enclosed in a vessel to simulate the shell side of heat exchanger. The cold fluid enters the tube shell through the top connection and right to left horizontal direction. The hot fluid flow through the no of tubes from top and discharge through the bottom opening. The pipe and the baffle are welded to a top flange in such a way that it provides the necessary flow through the tube and shell side of the test section and the required instrumentation A tank with electrical heaters is provided to heat the water to be circulated through the HE. There are three heaters, with a total power of 5000W. A controller is provided to maintain the temperature of fluid at the inlet of the test section at the set value. The hot fluid from the tank is pumped through the test section using a centrifugal pump of 1/2 hp power rating. Flow rate of hot fluid is measured using a rota meter. Both inlet and outlet temperatures of the hot fluid are measured by using Resistance Temperature Detectors and the values are available on digital displays. The property of fluid and heat exchanger material is given below in table No. 1 and 2

Table No. 1 Properties of Working Fluid

Parameter	Symbol	Unit	Cold Water (Shell)	Hot Water (Tube)
Fluid			Water	Water
Specific Heat	CP	KJ/Kg.K	4.178	4.178
Thermal Cond	K	W/m.K	0.615	0.615
Viscosity	µ	Kg/m.s	0.0013	0.0013
Prandtl 's No.	Pr		5.42	5.42
Density	ρ	Kg/m3	998.2	998.2

Table No. 2 Property of Material of Heat Exchanger

.

Property	*Steel*	*Copper*	*Brass*
Density (kg/m3)	7.87E+03	8.96E+03	8.60E+03
Poisson's Ratio	0.27-0.30	0.34	0.331
Thermal Conductivity(W/m-K)	59.5	401	115
Specific Heat (J/kg-K)	481	385	375

Figure 1 experimental setup of Shell and tube type heat exchanger

METHODOLOGY

In these research works we analyse the performance of heat exchanger by using experimental and Computational fluid dynamics so we categories these work in two different phase. In phase I, we perform the experimental analysis on physical model of shell and tube type heat exchanger and in Phase II, we perform the CFD analysis by creating virtual model in ansys modeller environment and browse these models in CFD workbench of Ansys 15 to perform the analysis.

A heat exchanger can be designed by the LMTD when inlet and outlet conditions are specified. When the problem is to determine the inlet and outlet temperatures for a particular heat exchanger, the analysis is performed more easily by using a method based on effectiveness of the heat exchanger and number of transfer units (NTU). The heat exchanger effectiveness is defined as the ratio of actual heat transfer to the maximum possible heat transfer.

Table 3: Calculation Table for All Velocity by Experimental method

S.No.	*Cold Water*			*Hot Water*			*Lmtd*
	Cold Water Velocity m/s	*Inlet Temp.C*	*Outlet Temp. C*	*Hot Water Velocity m/s*	*Inlet Temp. C*	*Outlet Temp. C*	
1	1	25	37	0.5	60	45	21.47
2	1.5	22	35	0.75	62	47	25.99
3	2	27	35	1	61	38	17.44
4	2.5	28	36	1.2	67	40.5	20.37
5	2.65	24	39.6	1.4	68	45.5	24.79
6	1.8	25	32	0..3	70	52	32.19
7	1.2	23	42	0.22	60	50	22.20

8	1.25	27	40	0.25	63	42	18.72
9	1.1	26	46	0.35	65	44	18.50
10	1.9	25	40	0.4	68	46	24.33
11	2.2	29	39	1	67	48	23.21
12	2.4	30	37	0.8	72	55	29.72

7. COMPUTATIONAL FLUID DYNAMIC ANALYSIS

In this project work a Computational Fluid dynamic Analysis has been performed in the assembly of shall and tube type Heat Exchanger. After the successful experiment of heat exchanger we move on the CFD analysis of heat exchanger. In order to perform the CFD analysis we need to create the model of heat exchanger and convert it into virtual environment. We had taken several measurement of physical setup of heat exchanger and taken the mean of measurement for modelling and finally we create the CAD model of heat exchanger in Ansys workbench.

Modelling of Shell Geometry

The typical modelling process is performed by the ANSYS 15.0 workbench. We are using the design modeller workbench for modelling of shell and tube geometry in ANSYS 15.0. There are two different parts in the assembly of heat exchanger in which one is cylindrical wall shell which contains the cold water and other is tube type geometry with dome in both end.

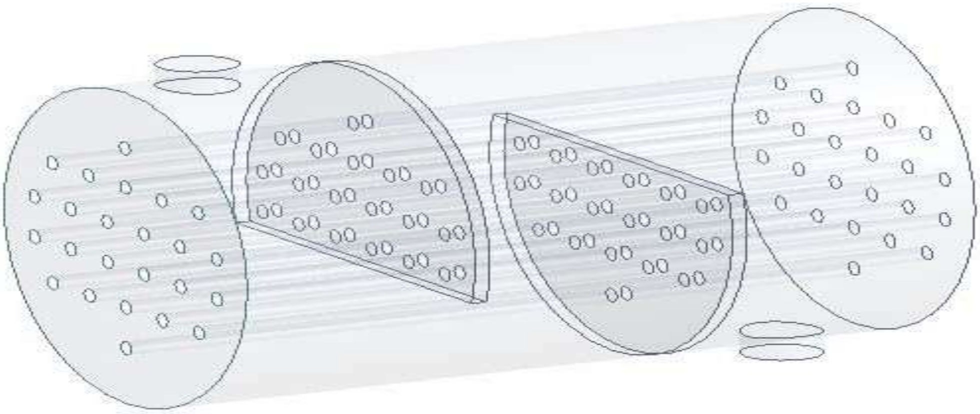

Figure 2: Model of Heat Exchanger Shell

Meshing of Heat Exchanger Model

FIGURE 3: Meshing Of Helical coil Heat Exchanger

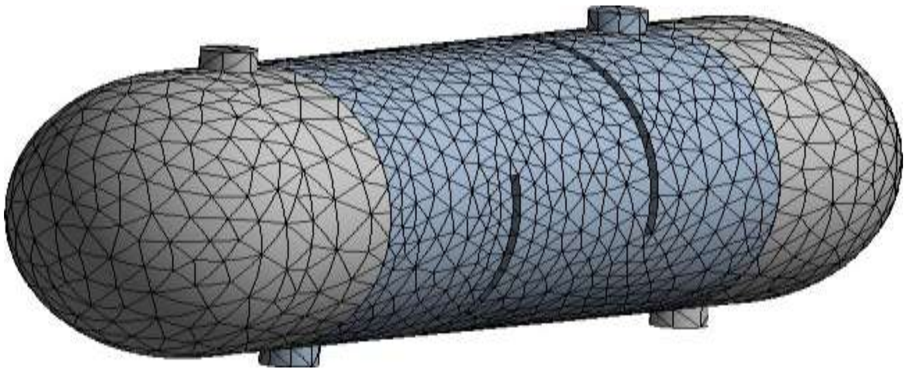

Table 4: Meshing Information of Heat Exchange

S.No	Meshing Information	Quantity
1	Number of Nodes	173016
2	Number of Elements	101185
3	Tetrahedra	305577
4	Wedges	35451
5	Pyramids	3622
6	Hexahedra	12926

Table 5: Properties of Flowing Fluids

S No	Name Of Fluid	Boundary Condition Type	Value
1	Hot water	Inlet Velocity	0.5 m/s
		Inlet Temperature	65° C
		Initial Pressure	1 atm
2	Cold water	Inlet Velocity	1 m/s
		Inlet Temperature	25° C
		Initial Pressure	1 atm

To perform the thermal analysis of heat exchanger in ANSYS 15.0, we need to define the domain; we have selected two domains for this purpose.

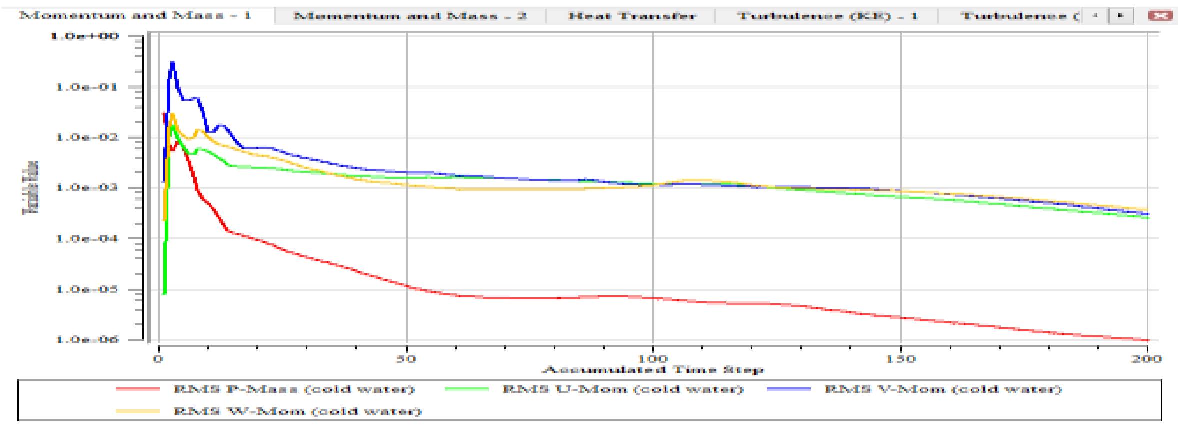

Figure 4: heat transfer graph of Helical Coil Heat Exchanger

8. RESULT & DISCUSSION

EXPERIMENTAL RESULT OF SHELL AND TUBE TYPE EXCHANGER

After the experimental procedure we get different set of result in the form of combination of inlet velocity and temperature for cold and hot fluid. The detail of inlet and out let temperature is given in the table below. After the critical analysis of the result we found that the heat transfer rate in increases due to increase the inlet velocity of the cold fluid and when we increase the velocity of hot fluid there is significant change in the temperature of outlet hot fluid. Similarly the value of LMTD is high for first two and last three experiment and for rest of experiment its nature is irregular. For experiment Number 6 and 8th the value of LMTD is almost same for both the combination of inlet velocity and temperature of hot and cold fluid.The overall effectiveness of the heat exchanger is lie in between the value of 0.51 to 0.67, which shows the performance of the heat exchanger and effectiveness is also vary according to the combination of velocity and inlet temperature of cold and hot fluid.

Table 6 Experimental Result of Shell & Tube Type Heat Exchanger For Counter Flow

S.No.	*COLD WATER*			*HOT WATER*				
	cold water velocity m/s	*Inlet Temp.C*	*Outlet Temp. C*	*hot water velocity m/s*	*Inlet Temp.C*	*Outlet Temp. C*	*LMTD*	*Effectiveness of HE*
1	1	25	30	0.5	60	40	21.64	0.57
2	1.5	22	35	0.75	62	42	23.33	0.50
3	2	27	45	1	61	40	14.45	0.62
4	2.5	28	48	1.2	67	41	15.81	0.67
5	2.65	24	49	1.4	68	40	17.46	0.64
6	1.8	25	44	0..3	70	41	20.60	0.64
7	1.2	23	50	0.22	60	41	13.61	0.51
8	1.25	27	40	0.25	63	45	20.40	0.50

9	1.1	26	46	0.35	65	39	15.81	0.67
10	1.9	25	38	0.4	68	41	21.96	0.64
11	2.2	29	45	1	67	50	21.50	0.45
12	2.4	30	49	0.8	72	51	21.98	0.50

CFD RESULT OF SHELL AND TUBE TYPE EXCHANGER

According to the experimental result we create the same combination of hot and cold fluid of the heat exchanger and perform each experiment in CFD workbench. There are 12 different separate combinations of inlet and out temperature of hot and cold fluid and different velocity. After the analysis of heat exchanger in CFD work bench we get set of output in the form of cold and hot outlet of both fluids which is given in the table 7 below.

Table 7 CFD Result of Shell & Tube Type Heat Exchanger For Counter Flow

S.No.	*Cold Water*			*Hot Water*			*LMTD*	*Effectiveness*
	velocity m/s	*Inlet Temp.C*	*Outlet Temp. C*	*velocity m/s*	*Inlet Temp.C*	*Outlet Temp. C*		
1	1	25	44	0.5	60	33	11.54	0.77
2	1.5	22	49	0.75	62	41	10.30	0.72
3	2	27	50	1	61	34	8.85	0.79
4	2.5	28	47	1.2	67	37	13.78	0.77
5	2.65	24	46	1.4	68	34	15.22	0.77
6	1.8	25	45	0..3	70	33	14.92	0.82
7	1.2	23	41	0.22	60	31	12.72	0.78
8	1.25	27	45	0.25	63	35	12.33	0.78
9	1.1	26	47	0.35	65	31	10.15	0.87
10	1.9	25	44	0.4	68	34	15.29	0.79
11	2.2	29	48	1.0	67	38	13.38	0.76
12	2.4	30	51	0.8	72	39	14.16	0.79

9. CONCLUSION & FUTURE SCOPE

Two different approaches have been performed to know the distribution of temperature and effectiveness of heat exchanger under the same working condition. The phase I we perform the experimental study in shell and tube type heat exchanger and taking 12 different combination of temperature and inlet velocity. Based on the output of experiment result we calculate the effectiveness and LMTD value for each experiment. Similarly in phase II we are using the same combination of velocity and temperature of hot and cold fluid and performed the CFD analysis. A comparative graph of distribution of LMTD and effectiveness i given in the figure 5 and 6.

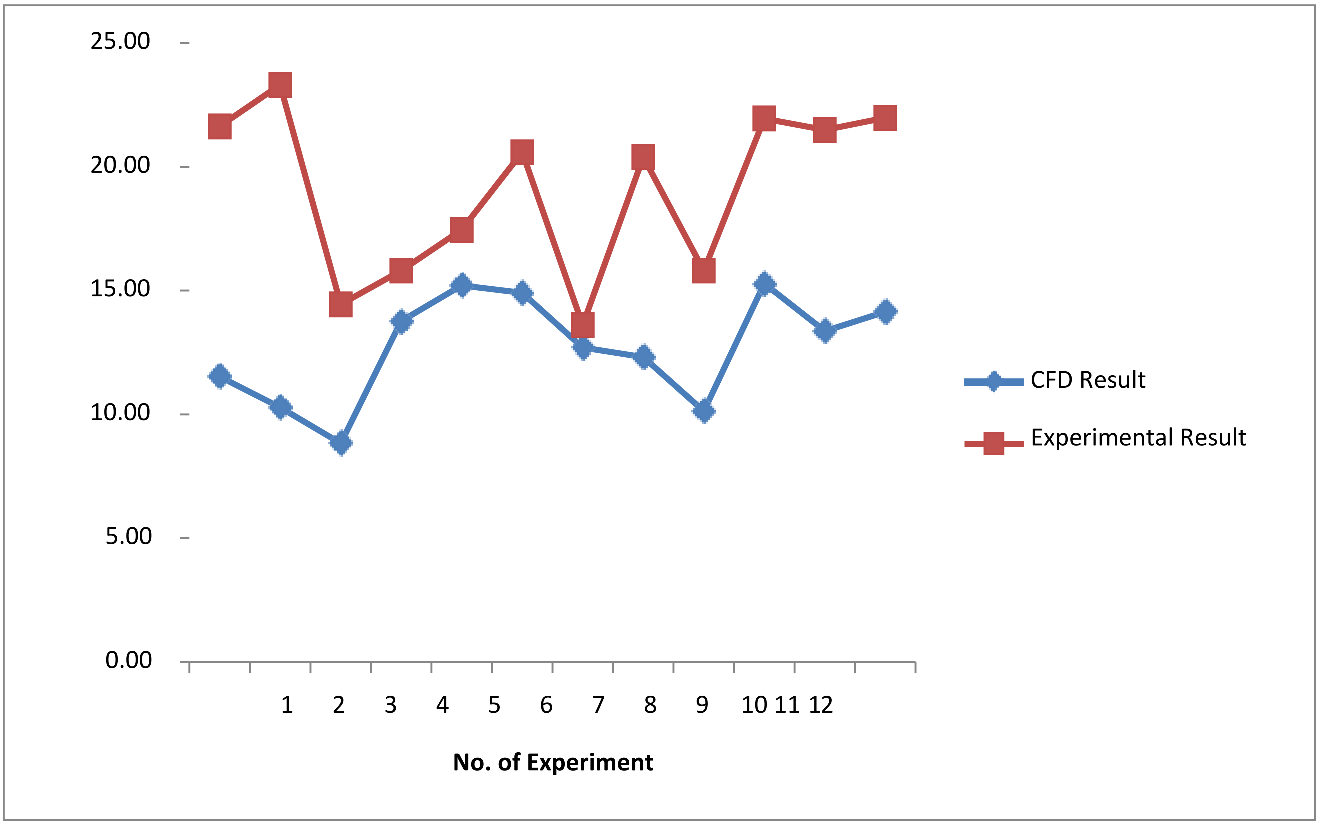

Figure No.5 comparative study of LMTD of CFD and Experimental Result

With reference to the figure no 5 we observed that the value of LMTD is higher for experimental study as compare to the CFD result. The nature of graph is irregular for first two experiments for both cases and its goes up for next three experiments. After that its nature is again irregular due to the velocity of inlet of hot and cold fluid. The experiment no seven shows same value for CFD and Experimental cases and its near about 13. For the first and last experiment the value of LMTD show significant difference between the CFD and experimental result.

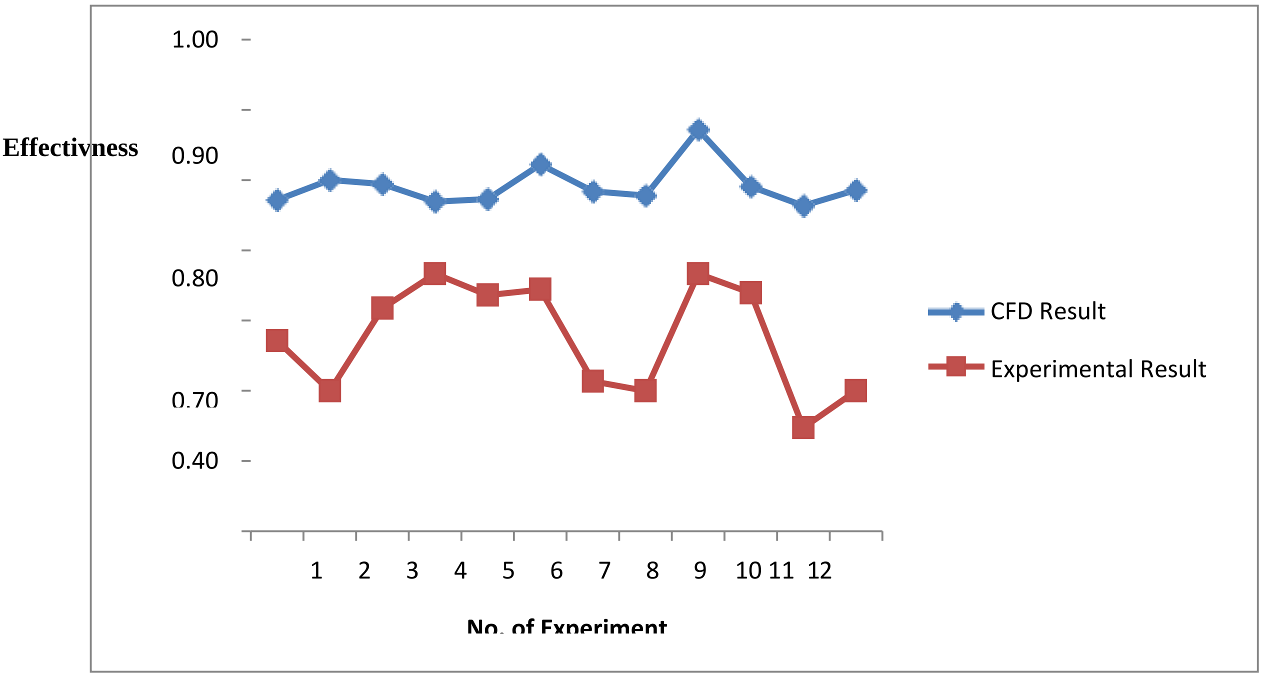

Figure No.6.comparative study of Effectiveness of CFD and Experimental Result

Figure no 6 show the value of CFD and experimental result of heat exchanger and we found that the effectiveness of CFD result is higher than the experimental result. The reason behind that is CFD workbench is work under the ideal condition and experimental study work on physical condition. The nature of graph is almost same for both the study and variation found during the experiment due to the change in velocity of fluid. The experiment Number nine shows large value of effectiveness both cases while the experiment no 11 shows smallest value of effectiveness for both cases. The inlet velocity of experiment 9 is 1.1 m/s and 2.2 for experiment no 11 for cold fluid while the velocity of hot fluid maintain at 0.35 m/s. Based on the analysis of the graph we concluded that the CFD result show better result as compare to the experimental result.

Future Scope

- The present work is concerned with result and analysis of heat transfer of heat exchanger with given specification it can also extended by changing the parameter of internal and external size of heat exchanger.
- In the present work water is used as working fluid. The work can be extended for different fluid.
- The present work is concerned with turbulent flow. The work can be further extended for different Reynolds no.
- Based on the 12 experiment of heat exchanger this work is also further extended by using the design of experiment and optimization by using the different algorithm.
- Heat exchanger is one of the most important equipment of the industry and already many researchers worked on that but still there having scope of engineering, in our work we can use different number of passes of pipe flow, and nature of flow like parallel and cross flow and we can also change the path of fluid.
- Current research work based on copper and brass pipe that can also change by other materials.

REFERENCES

[1] Dahare N B. “Experimental Analysis and Performance Characteristic Of Heat Transfer In Shell and Twisted Tube Heat Exchanger.” International journal of Engineering and Technology, 07, 1142-1148, 2015.

[2] Bhatt, D et all. “Shell and Tube Heat Exchanger Performance Analysis”, International Journal of Sciences and Research, 42, 1872-1881, 2012.

[3] Thavamani J, at all., “ Experimental and CFD analysis of Shell and tube heat exchanger”, Journal of Chemical and Pharmaceutical Sciences, 10, 84-86. 2017

[4] Guihong Pei et all“Heat transfer analysis of underground U type heat exchanger of ground source heat pump system,” Springer plus, 5, 1-15, 2016.

[5] Muhammad Mahmood Aslam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, Sarfaraz Khan, CFD applications in various heat exchangers design: A review, Applied Thermal Engineering 32 (2012) 1-12.

[6] Patel P K , Experimental “experimental and CFD analysis of tube helical coil heat exchanger”, International Journal of Advance Research and Innovative Ideas in Education ISSN: 2395-4396 1708-1718,2017

[7] R. Beigzadeh, Experimental and CFD Study of the Tube Configuration Effect on the Shell-Side Thermal Performance in a Shell and Helically Coiled Tube Heat Exchanger Iranian Journal of Chemical Engineering/ vol. 12 No.2/13-25,

[8] Pramod S. Purandare, Mandar M. Lele, Rajkumar Gupta, Parametric Analysis of Helical Coil Heat Exchanger, International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 1 Issue 8, October – 2012.