International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 18 (2018) pp. 13727-13733

Heat Exchanger Design Modification for Performance Optimization Using CFD Tools

Srinivasan S, M.E[1], Seenivasaperumal V, M.E[2], Manikandan G, M.E[3]

[1]*Assistant Professor, Mechanical Department, SSM Institute of Engineering and Technology, Dindigul, Tamil Nadu, India.*

[2]*Assistant Professor, Mechanical Department, INFO Institute of Engineering and Technology, Coimbatore, Tamil Nadu, India.*

[3]*Assistant Professor, Mechanical Department, Kongu Engineering College, Erode, Tamil Nadu, India.*

Abstract

Heat exchangers are used in the thermal system to maintain the temperature of the working liquid. Among the various types of heat exchanger, the shell and tube heat exchanger is the most commonly used heat exchanger based on its simple design and performance aspects. Even though these shell and tube heat exchangers operates at its designed point, it can be even effectively designed to achieve better heat transfer rate.

Since most of the shell and tube heat exchangers are designed based on the traditional design concepts, in this paper, we have planned to modify the design of the heat exchanger. For this purpose, we have selected a reference heat exchanger with its practical performance results. Using the design data from the existing heat exchanger, the CAD model was generated using solid works software and it was analysed using the CFD software under the actual operating conditions.

Then, the design modifications was carried out in the inner tubes and baffles accordingly. Initially we have changed the design of baffles and based on the analysis results, the cross section of the tube was modified. These modified design was analysed using the CFD tools under the same operating conditions and the results was compared with the actual and modified designs.

By this, we will be justifying the application of CFD tools in the design of heat exchangers to predict its performance in the early design stage. Due to this, the time and money invested on the man and equipments will be reduced to the industry for developing an efficient heat exchanger.

Keywords: Heat exchangers – CFD tools – performance analysis – design optimization.

INTRODUCTION

Heat exchangers are widely used in the thermal systems where there are requirement of maintaining the system temperature in order to get the quality product. The quality of the final product is directly associated with the efficiency and effectiveness of the heat exchanger. The efficiency of the heat exchanger is based on the quantitative values of the inlet and exit temperature difference.

The efficiency of this type of heat exchanger is determined by the amount of heat transferred from hot to cold mediums and in vice versa. Also in some peculiar cases, for the customized purposes, the flow inside the heat exchangers are changes from counter flow to parallel flow. This is purely based on the application where it is been used. In most of the cases, the type of fluid flowing in the shell and in the tubes are also changed to get better efficiency.

Based on the above, this project is based on the performance study of a shell and tube heat exchanger using a CFD tool. This study is carried out with respect to the available practical data. The practical data are taken from the reference paper and the design of the heat exchanger is also taken from the reference paper. The reference paper contains the practical experimental data. This study is to justify the application of CFD tools in the design of the heat exchanger system. The analysis will be carried under varied conditions as per the reference paper.

The first phase of this project deals with the study and analysis of the heat exchangers along with the selection of reference paper continued by the modelling of the heat exchanger system. The second phase of this project will be contained with the CFD analysis of the system using the solid works flow simulation tools. Then the experimental and CFD results will be compared to justify the effective application of the CFD tools in thermal system design.

METHODOLOGY

Methodology is the basic requirement for a project, because it defines the proper start and end conditions of the works to be done. Proper planning and execution of the workflow decides the successful completion of the project. The methodology of this project is as follows.

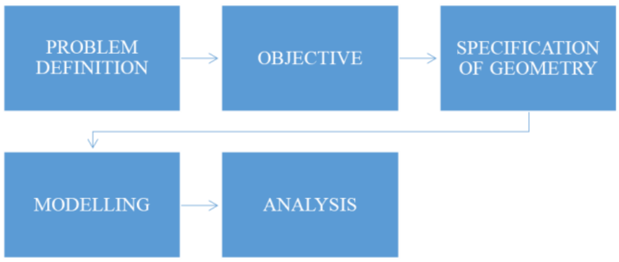

Figure 1: The strategic planning process

Problem Definition

The problem is defined as to model a heat exchanger as per the physical dimensions available in the industry and to conduct the CFD analysis for the existing operating conditions of 55 degree Celsius and 25 degree Celsius with hot water in shell and cold water in tube. This analysis have to be done under parallel flow conditions and verify the CFD results are same as actual experimental results with general limitations.

Also the analysis should also be conducted for various operating conditions like parallel flow and counter flows and varied baffle geometric conditions.

Objective

The objective of this project is to justify the applications of CFD tools in the thermal system design. Also the baffle conditions have to be optimized accordingly to find the best operating conditions.

Geometric Specifications

The geometry specifications are taken from the design data available in the reference journals [1]. The following are the details of the heat exchanger used for analysis.

Figure 2: Real time experimental setup

Shell Details

Outer diameter = 142mm

Inner diameter = 136mm

Length of the HE = 1500mm

No. of baffles = 5

Distance between baffles = 300mm

Baffle opening = 25% (except first and last)

Tube Details

Outer diameter = 23mm

Inner diameter = 20mm

Length = 1200mm

No. of tubes = 9

Material of construction

Shell = Stainless Steel

Tubes = Copper

Baffles = Copper

Modelling

The following are the CAD images of the Heat exchanger modelled using Solid Works and drafter using Auto CAD.

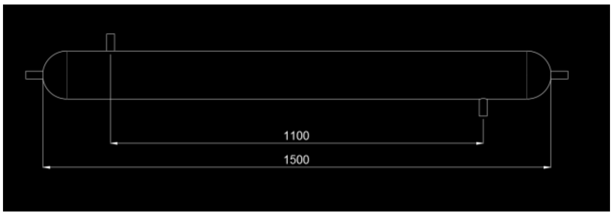

Figure 3: Dimensions of Shell and openings

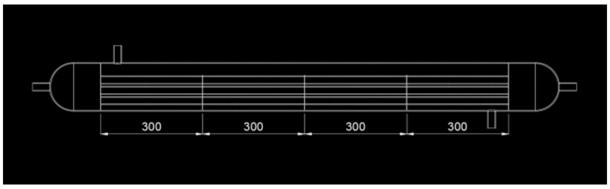

Figure 4: Dimensions of baffle arrangements

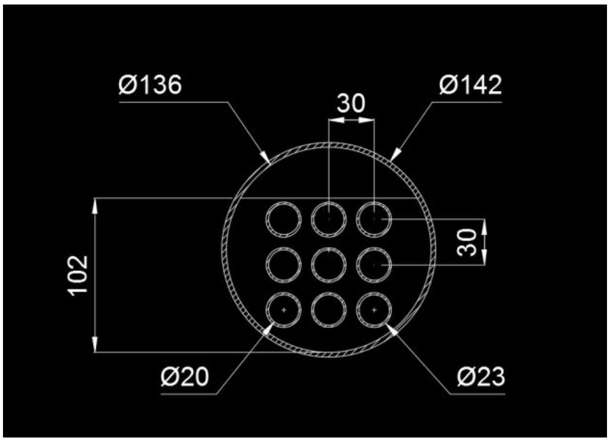

Figure 5: 8 Dimensions of shell, tube and baffle

Figure 6: Front view of the Heat Exchanger

International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 18 (2018) pp. 13727-13733

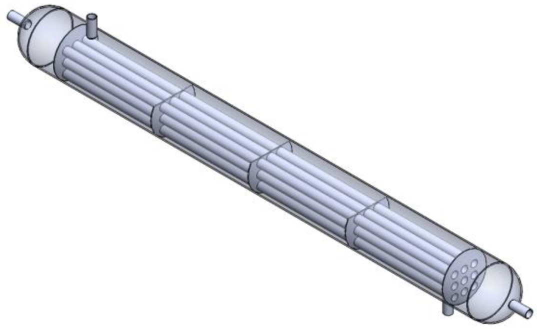

Figure 7: View of tubes and baffles

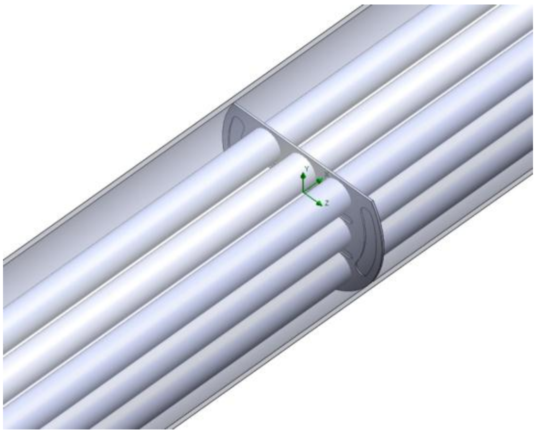

Figure 8: Slots in baffles (slots are of half the thickness)

Analysis

The performance analysis for the heat exchanger was done using Solid Works Fluid Flow Simulation. The analysis was conducted for the following operating conditions.

Case 1: Hot water in Shell and Cold water in Tube – Parallel flow (actual condition)

Case 2: Hot water in Shell and Cold water in Tube – Counter flow

Case 3: Hot water in Tube and Cold water in Shell – Parallel flow

Case 4: Hot water in Tube and Cold water in Shell – Counter flow

Among these analysis, the case 1 is the actual condition in which the system is been operated and tested in the industry. The following are the results of the experimental test.

Table 1: Ranked problems of the Russian road transport enterprises and options for solution of these problems

Dated- 11th & 12th of April 2014 (12:00 to 4:30 PM), ambient temperature: 33 (°C)

Serial number	Hot water inlet (°C)	Hot water outlet (°C)	Cold water inlet (°C)	Cold water outlet (°C)	Degree of cooling (°C)	Degree of heating (°C)	Effectiveness of heat exchanger
1.	55.3	47.5	25.5	31.1	7.8	5.6	0.26174
2.	55.4	47.9	24.2	30.3	7.5	6.1	0.24038
3.	55.0	47.5	25.0	30.0	7.5	5.0	0.25
4.	55.3	47.5	25.5	31.1	7.8	5.6	0.26174
5.	55.4	47.9	24.2	30.3	7.5	6.1	0.24038
6.	55.7	48.1	24.7	30.0	7.6	5.3	0.24516
7.	55.3	48.3	25.2	29.3	7.0	4.1	0.23255

Form the above set, the set 3 is taken for CFD analysis and for comparison purpose. The above set is for case 1 analysis. The analysis results of all the four cases will be compared and the best operating conditions will be selected for the further development activities. The following are the proposed development models.

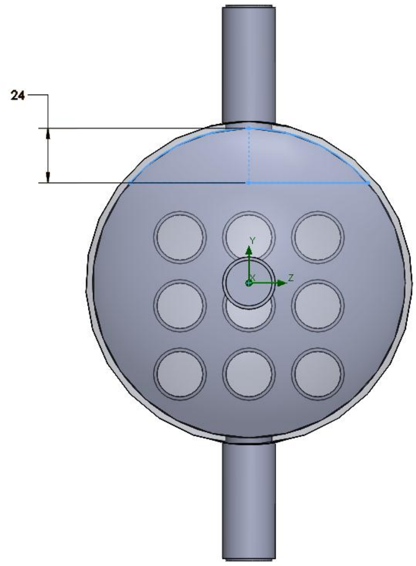

Figure 9: Baffle gap reduced from 34 mm to 24mm

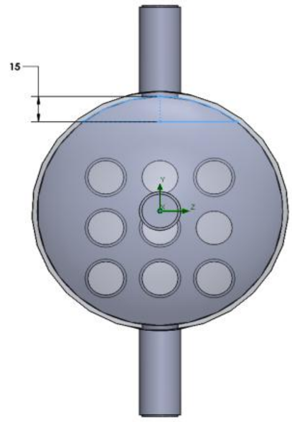

Figure 10: Baffle gap reduced from 34 mm to 15mm

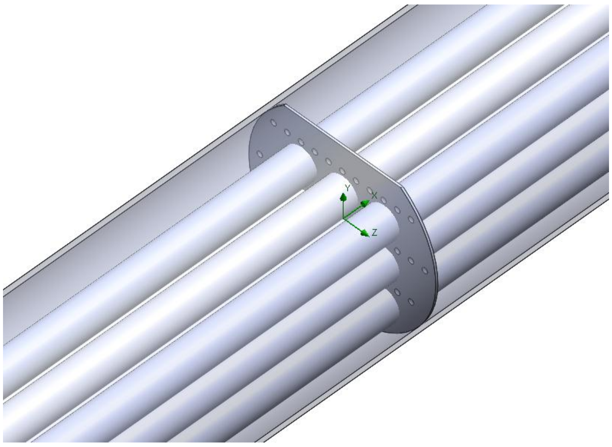

Figure 11: Baffle gap 15mm along with spherical dimples on baffles

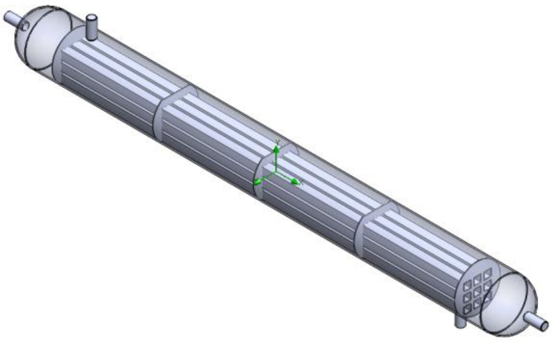

Figure 12: Proposed change in tube shape _ Rectangular shape tubes

This rectangular shaped tube design will be implemented in the design which is having better performance among those which have been mentioned earlier.

The CFD analysis was conducted under the following 3 steps.

- Pre-processing
- Solution
- Post-processing

The model importing and cleaning, meshing, boundary conditions and material property assigning are all done at the pre-processing stage. The solver settings and output settings and simulations are carried at the solution stage. The extraction of results from the saved database in the form of contour plots and tabulated values are done in the post-processing and this post-processing will be explained in the upcoming chapter.

The boundary conditions used for the analysis are as follows.

1. Hot liquid inlet = 55 0C
2. Cold liquid inlet = 25 0C
3. Mass flow rate (Hot) = 0.027 Kg/sec
4. Mass flow rate (Cold) = 0.014 Kg/sec
5. Ambient temperature = 30 0C
6. Hot and cold liquid = Water

Assumptions made are:

1. Flow is laminar and turbulent
2. U = 750 W/m2K (based on data available in net)

RESULTS AND DISCUSSION

The following are the typical CFD contour plot outputs. These outputs are retrieved using the post processing tools. In total 9 cases have been analyzed and its quantitative results are displayed as tabular columns and histogram charts following the below schematic CFD plots.

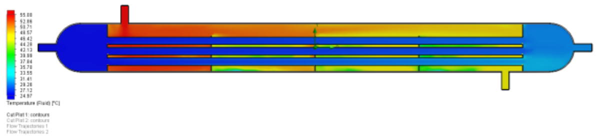

Figure 13: Interior fluid temperature (front plane cut section)

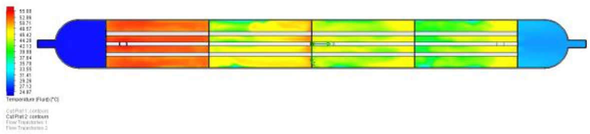

Figure 14: Interior fluid temperature (top plane cut section)

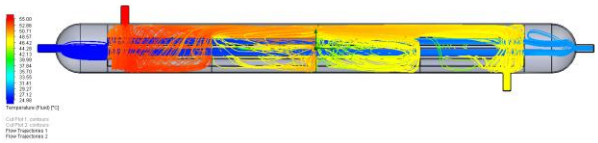

Figure 15: Interior fluid flow pattern (flow trajectories)

Table 2: Results of Case 1 to Case 4 (actual model with varied operating conditions)

CONDITIONS	HOT OUTLET [°C]	COLD OUTLET [°C]
CASE 1	46.845	29.393
CASE 2	46.689	29.594
CASE 3	48.522	28.913
CASE 4	48.473	29.201

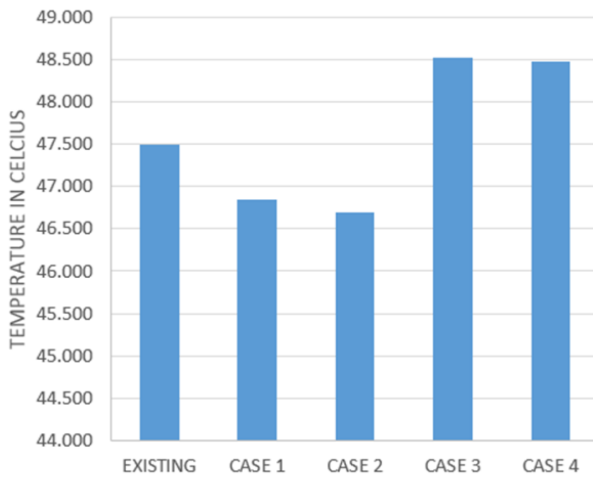

Figure 16: Hot water outlet temperatures – case 1-4

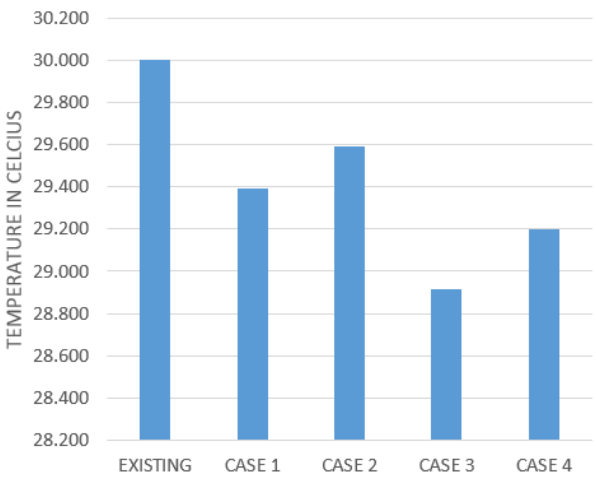

Figure 17: Cold water outlet temperatures case 1-4

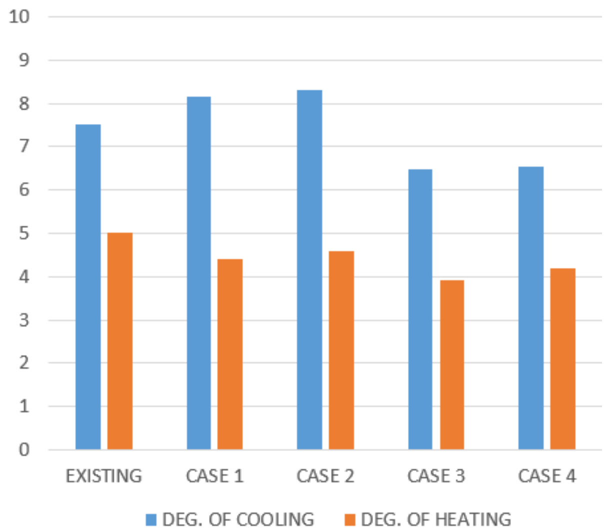

Figure 18: Degrees of cooling and heating – case 1-4

Table 3: Results of Case 5 to Case 8 (geometry modifications with case 2 condition)

CONDITIONS	HOT OUTLET [°C]	COLD OUTLET [°C]
CASE 5	46.838	29.600
CASE 6	46.414	29.675
CASE 7	45.913	29.869
CASE 8	45.745	29.885

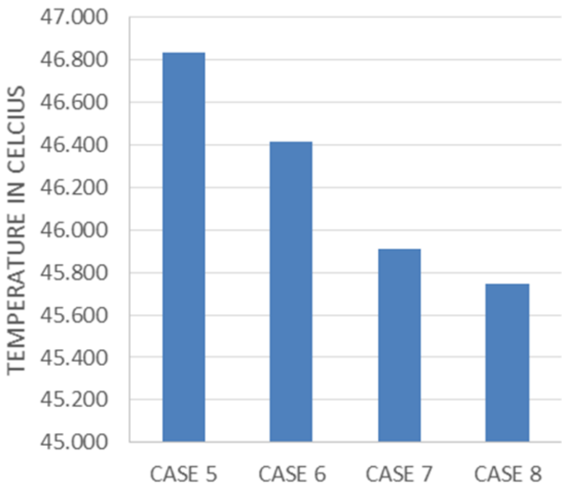

Figure 19: Hot water outlet temperatures – case 5-8

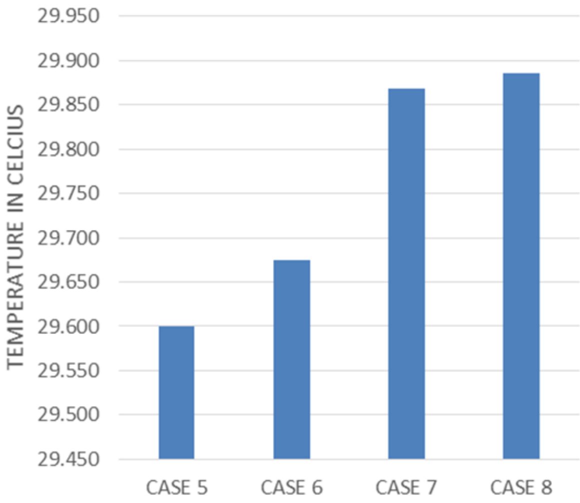

Figure 20: Cold water outlet temperatures case 5-8

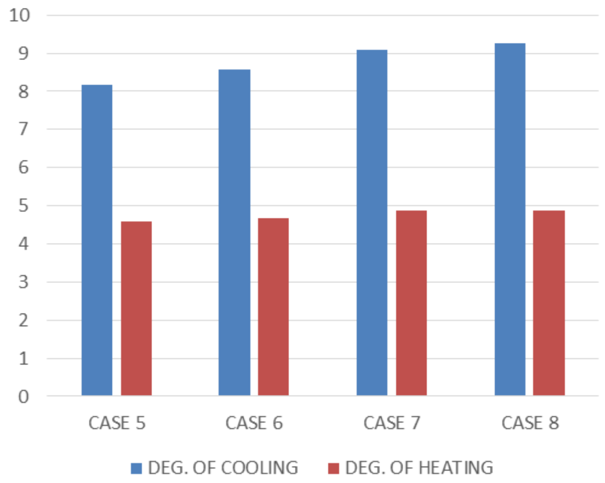

Figure 21: Degrees of cooling and heating – case 5-8

From the actual experimental data, the third serial data was taken for analysis and the obtained results are 47.5 and 30.0o C, whereas for the same data, the CFD results obtained are, 46.845 and 29.393o C (case 1). This proves that, the CFD results are comparatively matching with the experimental results. Hence on further analysis, it was found that the case 2 operating condition was having better results than the other three cases.

Using the case 2 operating conditions, the geometry of the heat exchanger was modified and analysed. On observation it was found that, the geometry with case 2 and slots is having slight better performance. On further improvement activity, the baffle gap was reduced and analysed. This shows that the performance increases as the volume of water inside the shell increases, but not beyond a limit as it affects the discharge issues. So the gap of 15mm was maintained and for this condition, the baffles are analysed for spherical dimples instead of slots. This condition results proves that the output is much better than the other operating conditions. The spherical dimples are used as an innovative idea.

Based on the above results, it was found that the modification of baffles with spherical dimples are having better performance than all other cases. In addition to this, an attempt was made to check the performance by changing the tube's cross section to rectangular from circular. The same was done and its performance results are plotted below.

Table 4: Results of Case 8 and Case 9

CONDITIONS	HOT OUTLET [°C]	COLD OUTLET [°C]
CASE 8	45.745	29.885
CASE 9	45.363	30.377

From the above it can be said that, the change in cross section of the tube is having considerable amount of impact in the performance of the system. On comparing the results, the proposed rectangular shape along with the spherical dimples is having better performance than the other cases.

CONCLUSION

The requirement of the industry was to analyse their heat exchanger for the using CFD tools and compare the results and if the results are satisfied, then the development of the system have to be carried out. This was taken as the problem statement and the actual setup was modelled and analysed under the actual boundary conditions to validate the CFD results. The CFD outputs have justified that the actual results and software results are mere co-incidence based on some assumptions.

Further the operating conditions of the system was varied and analysed to find the best effective condition. The case 2 condition was found to be the best ant it was suggested. Now based on this case 2 results, the modifications in baffle design were conducted and it was analysed under same boundary conditions. The baffle designs with spherical dimples was found to be having better performance compared to the other design suggestions.

Also, an initiative was made to check the performance by changing the cross section of the tubes for the design having better performance. This was implemented using the rectangular cross sections for the tube and the analysis was conducted and the results were found to be better than the all other proposed cases.

Hence, it can be concluded that, the rectangular tube is having better performance than the circular tube and the modifications in the baffle designs can be implemented to get the better performance.

REFERENCES

[1] Vindhya Vasiny Prasad Dubey, Raj Rajat Verma, Piyush Shanker And A.K.Srivastava, "Performance Analysis Of Shell & Tube Type Heat Exchanger Under The Effect Of Varied Operating Conditions", Journal Of Mechanical And Civil Engineering (Iosr-Jmce), Volume 11, Issue 3 Ver. Vi (May- Jun. 2014), Pp 08-17.

[2] Vikas Kumar, D. Gangacharyulu, Parlapalli Ms Rao And R. S. Barve, "Cfd Analysis Of Cross Flow Air To Air Tube Type Heat Exchanger".

[3] 3. Avinash D Jadhav, Tushar A Koli, "Cfd Analysis Of Shell And Tube Heat Exchanger To Study The Effect Of Baffle Cut On The Pressure Drop", International Journal Of Research In Aeronautical And Mechanical Engineering, 2321-3051i.

[4] 4. Hetal Kotwal, D.S Patel, "Cfd Analysis Of Shell And Tube Heat Exchanger- A Review", International

Journal Of Engineering Science And Innovative Technology, Volume 2, Issue 2, March 2013

[5] Alok Vyas, Mr. Prashant Sharma, “An Experimental Analysis Study To Improve Performance Of Tubular Heat Exchangers”, Alok Vyas Et Al Int. Journal Of Engineering Research And Applications, Vol. 3, Issue 6, Nov-Dec 2013, Pp.1804-1809.

[6] Apu Roy, D.H.Das, “Cfd Analysis Of A Shell And Finned Tube Heat Exchanger For Waste Heat Recovery Applications”, International Journal Of Mechanical & Industrial Engineering, Volume-1 Issue-1, 2011