CFD Analysis of Shell and Tube Heat Exchanger using triangular fins for waste heat recovery processes

Swapnaneel Sarma
Mechanical Engineering Department
NIT Silchar
Silchar, India

D.H.Das
Mechanical Engineering Department
Associate Professor, NIT Silchar
Silchar, India

Abstract—**The energy present in the exit stream of many energy conversion devices such as I.C engine gas turbine etc. goes as waste, if not utilized properly. The present work has been carried out with a view to predicting the performance of a shell and finned tube heat exchanger in the light of waste heat recovery application. The performance of the heat exchanger has been evaluated by using the CFD package fluent 6.3.26 and has been compared with the existing experimental values. An attempt has also been made to calculate the performance of the above heat exchanger by considering triangular fins instead of regular rectangular fins and the result so obtained have been compared. The performance parameters pertaining to heat exchanger such as effectiveness, overall heat transfer coefficient, energy extraction rate etc., have been reported in this work.**

I. INTRODUCTION

The recovery of waste heat from exhaust gases has become essential due to the declining energy resources and production cost. The need to use energy more efficiently has become a necessity since the large increase in oil prices. Energy preservation is primarily concerned with the task of extracting maximum production from specific energy expenditure. A major result of the energy conservation drive is the development of process recovery aimed at reducing the amount of waste heat discharged to the environment thus increasing the overall efficiency of various processes and systems. Heat recovery conserves energy, reduces the overall operating costs and thereby reduces peak loads.

The core of a heat recovery system is the heat exchanger. Decreasing size and increasing heat load is the typical feature of the modern day Heat Exchanger. Finned surfaces have more heat transfer surface due to their compact configuration. Compact heat exchanger can be used for heat recovery purpose because of its several advantages over conventional type. Compact heat exchangers are characterized by having a high area density, which means a high ratio of heat transfer surface to heat exchanger volume. Finned tube heat exchanger comes under the category of compact heat exchanger due to its compact configuration. The waste gas from various sources at different temperature ranges are usually dumped into the environment which could otherwise be used for some useful purposes.

II. OBJECTIVES OF THE PRESENT WORK

The present work begins with the CFD analysis of shell and finned tube heat exchanger in order to see the effect of temperature rise and pressure drop along the length of the finned tube and the shell. The exhaust gas from diesel engine has been used as the shell side fluid for heat transfer analysis. The exhaust gas transfer heat to the cold fluid (water) that is flowing through the tube.

CFD provides the flexibility to change design parameters without the expense of hardware changes. It therefore costs less than laboratory or field experiments, allowing engineers to try more alternative designs than would be feasible otherwise. It also reduces design cycle time and cost by optimizing through computer predictions and provides higher level of confidence in prototype or field installed performance. Moreover it investigates and understands the "why" for existing problem or new equipment.

In the light of the above discussion, the present work has been taken up aiming at achieving the following objectives.

- To perform a CFD analysis of Finned tube heat exchanger using FLUENT 6.3.16 after modeling the heat exchanger with GAMBIT 2.4.14.
- To observe the effect on temperature rise and pressure drop along the length of the finned tube and shell.
- To perform CFD analysis of Finned Tube Heat Exchanger using water as Heat Transfer Fluid (HTF).

III. COMPUTATIONAL FLUID DYNAMICS

The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. One method is to discretize the spatial domain into small cells to form a volume mesh or grid and then apply a suitable algorithm to solve the equations of motion (Euler equations for inviscid andNavier-Stokes equations for viscous flow). In addition, such a mesh can be either irregular (for instance consisting of triangles in 2D or pyramidal solids in 3D) or regular; the distinguishing characteristic of the former is that each cell must be stored separately in memory. It is possible to directly solve the Navier-Stokes equations for laminar flows and for turbulent flows when all of the relevant length scales can be resolved by the grid (a direct numerical simulation). In general however, the range of length scales appropriate to the problem is larger than even today's massively parallel computers can model. In these cases, turbulent flow simulations require the introduction of a turbulence model. Large eddy simulations (LES) and the Reynolds-averaged Navier-Stokes equations (RANS) formulation, with the k-ε model or the Reynolds stress model, are two techniques for dealing with these scales. In many instances, other equations are solved simultaneously with the Navier-Stokes equations. These other equations can include those describing species concentration (mass transfer), chemical reactions, heat transfer, etc. More advanced codes allow the simulation of more complex cases involving multi-phase flows (e.g. liquid/gas, solid/gas, liquid/solid), non-Newtonian fluids (such as blood), or chemically reacting flows (such as combustion).

A. *Basic Approach to using CFD*

a) Pre-processor: Establishing the model

- Identify the process or equipment to be evaluated.
- Represent the geometry of interest using CAD tools.
- Use the CAD representation to create a volume flow domain around the equipment containing the critical flow phenomena.
- Create a computational mesh in the flow domain.

b) Solver:

- Indentify and apply conditions at the domain boundary.
- Solve the governing equations on the computational mesh using analysis software.

c) Post processor: Interpreting the results

- Post-process the completed solutions to highlight findings.
- Interpret the prediction to determine design iterations or possible solutions, if needed.

B. *Governing Equations*

Each CFD software package has to produce a prediction of the way in which a fluid will flow for a given situation. To do this the package must calculate numerical solutions to the equations that govern the flow of fluids. For the analyst, it is important to have an understanding of both the basic flow features that can occur and so must be modeled and the equations that govern fluid flow. The physical aspects of any fluid flow and heat transfer are governed by three fundamental principles.

- Continuity equation
- Momentum equation and
- Energy equation.

C. *Choosing a Turbulence model*

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. Hence keeping all this in mind the Standard *k-ε* Model was chosen.

IV. EXPERIMENTAL INVESTIGATION

The major criterion in the design of waste heat recovery system is the proper selection of heat exchanger with optimum conditions. In the present investigation, the objective is to extract heat from the exhaust gas. This could be achieved either by embedding the heat exchanger coil surface inside the storage tank where the storage material is present and allowed to pass the exhaust gas through the heat exchanger coil or providing a separate heat exchanger through which heat transfer fluid is circulated to extract heat from the exhaust.

A. *GEOMETRICAL MODELLING*

The geometric model of finned tube heat exchanger was made on Gambit. The heat exchanger specifications are as follows:

Table 1

1.	Shell outer diameter	323mm
2.	Shell Thickness	6 mm.
3.	Length of the shell	500 mm
4.	Tube outer diameter	12.5 mm
5.	Tube Thickness	1.65 mm
6.	Tube type	Finned tube
7.	Transverse Pitch	37.5 mm
8.	Longitudinal pitch	37.5 mm
9.	Type of Tube layout	Square layout
10.	Type of Tube arrangement	In line arrangement
11.	No of tube inside the shell	36
12.	Fin Thickness	2 mm
13.	Fin height	6 mm
14.	Type of Fin	Triangular fin

The shell material, fin and tube materials is mild steel, and copper respectively. Longitudinal type of fin tube has been used and inline square arrangement has been adopted. The girded view has shown in fig. 1 & fig.2 respectively.

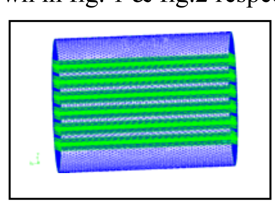

Fig. 4.1 Grided view of Heat exchanger

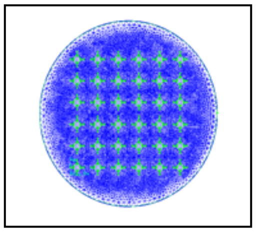

Fig. 4.2 C.S view of gridded heat exchanger

B. PROPERTIES OF THE WORKING FLUID

The model which has been developed on gambit is taken for further analysis. Here water is taken as heat transfer fluid. For shell side fluid exhaust gas from 15 HP diesel engine has been considered.

C. DATA COLLECTION AND ANALYSIS

The shell side fluid is the exhaust gas coming from a 15 hp engine at 120 ^{0}C. The tube side fluid temperature is taken as 25 ^{0}C. Keeping the shell side velocity constant and the tube side velocity was varied at various conditions.

D. Calculations

- Overall Heat Transfer Coefficient

The thermal resistance (R) can be expressed as an overall heat transfer coefficient based on either the inside or the outside surface of the tube. Overall heat transfer coefficient (U_o) based on outer surface is defined as

$$U_o = \frac{1}{\left(\frac{d_o}{d_i}\right)\left(\frac{1}{h_i}\right) + \left[\frac{1}{2k}\right] d_o \ln\left(\frac{d_o}{d_i}\right) + \left(\frac{1}{h_o}\right)}$$

Similarly, the overall heat transfer coefficient (U_i) based on inner surface is defined as

$$U_i = \frac{1}{\left(\frac{d_i}{d_o}\right)\left(\frac{1}{h_o}\right) + \left[\frac{1}{2k}\right] d_i \ln\left(\frac{d_o}{d_i}\right) + \left(\frac{1}{h_i}\right)}$$

When the wall thickness is small and its thermal conductivity is high, the tube resistance can be neglected and the overall heat transfer coefficient for inner surface reduces to

$$U_i = \frac{1}{\left(\frac{1}{h_i}\right) + \left(\frac{1}{h_o}\right)}$$

- Number of Transfer Unit (NTU)

Number of Transfer Unit (NTU) is defined as the ratio of overall thermal conductance to the smallest heat capacity rate.

$$NTU = \frac{U_m A}{C_{min}}$$

- Effectiveness (ε)

Under ideal condition, using the value of actual heat transfer rate (q) from the energy conservation equation, the effectiveness (ε_1) valid for all flow arrangement of the two fluids is given by

$$\varepsilon_1 = \frac{T_{h,i} - T_{h,o}}{T_{h,i} - T_{c,i}}$$

$$\varepsilon_2 = \frac{T_{c,o} - T_{c,i}}{T_{h,i} - T_{c,i}}$$

V. RESULTS AND DISCUSSION

The analysis has been carried out for five cases by varying the tube side fluid velocity from 0.85 m/s to 1.45 m/s. The shell side fluid velocity was kept constant at 0.0709 m/s. The purpose of CFD analysis was to see the effect of temperature rise and pressure drop in the finned tube with the increase in mass flow rate of the tube side fluid as well as to obtain the overall heat transfer coefficient and the effectiveness of the finned tube heat exchanger for counter flow configuration.

The temperature contour of the finned tube and shell are shown in fig.5.1 to 5.7. It is evident from the temperature contour of the shell that the temperature of the shell side fluid decreases after exchanging its heat with the fluid that is flowing through the finned tubes. Since the shell side fluid become mixed along the path of flow, therefore the exit temperature for the shell side fluid become nearly uniform. Pressure drop can be easily visible from the figure no 5.3 thereby making the necessity of large pumping power is requirement. Reducing the length may solve the problem to a large extent. The pressure drop of exhaust side is very low as observed from figures. It implies that there is no back pressure and hence there is no effect on working in the engine.

A. Analysis of the finned tube heat exchanger using water as heat transfer fluid

Case-I: Tube side fluid velocity 0.85 m/s, Shell side fluid velocity 0.0709 m/s

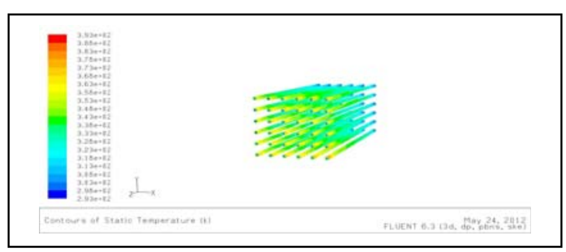

Fig 5.1.Temperature contour of tube (K)

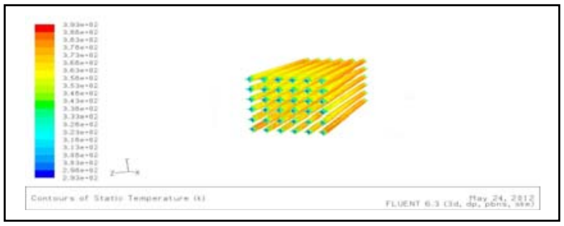

Fig.5.2 Temperature contour of fin surfaces (K)

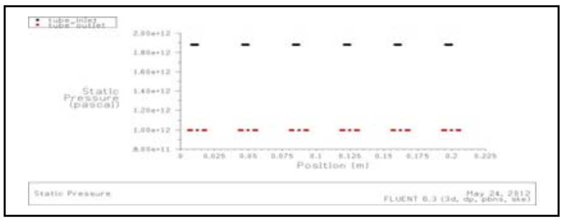

Fig.5.3 XY plot of pressure drop along the tube inlet and outlet (Pa)

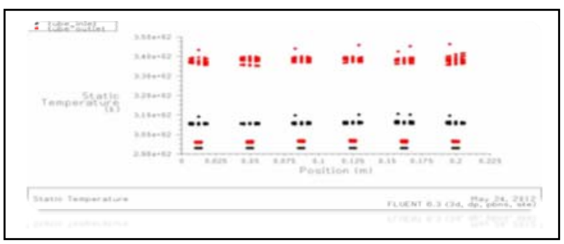

Fig.5.4 XY plot of Static Temperature of tube inlet and outlet

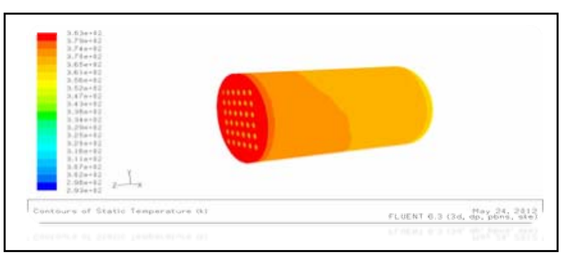

Fig.5.5. Temperature contour of the shell (K)

Case II: Tube side fluid velocity 1.0 m/s, Shell side fluid velocity 0.0709 m/s.

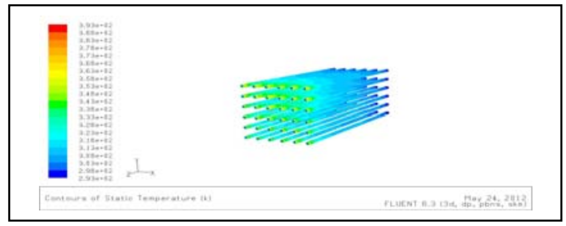

Fig:-5.6. Temperature contour of tube (K)

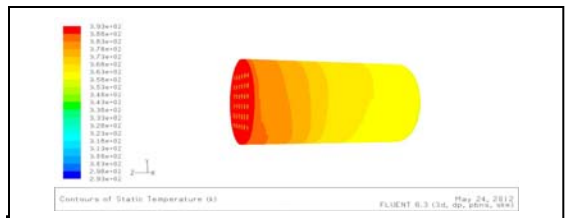

Fig.5.7. Temperature contour of the shell (K)

B. Data from fluent analysis.

Table 5.1 Tube side data

Sl. No	Velocity (m/s)	Inlet temp (K)	Outlet temp (K)	Mean temp (K)	Reynolds no	Prandtl no	Nusselt no
1.	0.85	298	336	317	10574.15	4.80	71.38
2.	1.00	298	334	316	12440.17	5.12	83.42
3.	1.15	298	333	315.5	14306.21	5.45	95.65
4.	1.30	298	331	314.5	16172.23	5.69	107.34

5.	1.45	298	330	314	18038.26	5.90	118.85

Table 5.2 Shell side data

SL. No	Tube fluid velocity (m/s)	Shell fluid velocity (m/s)	Shell inlet Temp (K)	Shell Outlet Temp (K)	Mean Temp (K)	Prandtl No	Reynolds No	Nusselt No
1.	0.85	0.0709	393	356	374.5	0.221	2427.40	7.46
2.	1.00	0.0709	393	354	373.5	0.227	2455.55	7.59
3.	1.15	0.0709	393	350	371.5	0.230	2467.78	7.65
4.	1.30	0.0709	393	342	367.5	0.234	2479.43	7.72
5.	1.45	0.0709	393	340	366.5	0.238	2484.56	7.77

Table 5.3.Overall Heat Transfer coefficient (W/m^2K)

Sl. No	Velocity (m/s)	Inner Heat Transfer Co-eff.(W/m^2K)	Outer Tube Heat Transfer Co-eff. (W/m^2K)	Friction factor.(f)	Overall Heat Transfer co-eff. (W/m^2K)
1	0.85	2226.25	2042.34	0.00779	317.75
2	1.00	2443.85	2342.50	0.00748	330.45
3	1.15	2764.30	2576.32	0.00722	343.85
4	1.30	2957.51	2897.45	0.00701	355.46
5	1.45	3219.46	3157.92	0.00681	363.80

Table 5.4 Heat capacity

SL. No	Cp of cold flow(J/kg K)	Cp of hot flow.(J/kg K)	Heat capacity of cold flow(W/K)	Heat capacity of hot flow(W/K)	Effectiveness (ε) (%)
1.	4182.00	440.27	10245.0	2013.14	31.1
2	4182.00	440.27	10245.2	2010.78	45
3	4182.00	440.27	10245.7	2006.42	50
4	4182.00	440.27	10245.2	2001.56	52
5	4182.00	440.27	10245.7	1994.17	58

Table- 5.5 Heat Extraction rate

Sl No	Tube fluid velocity(m/s)	Specific heat (J/Kg K)	Shell outlet temperature(K)	Heat extraction rate (%)	Sl No
1	0.85	440.27	356	25	1
2	1.00	442.27	354	30	2
3	1.15	445.27	350	37	3
4	1.30	447.27	342	40	4

C. Graphical representation of the obtained results

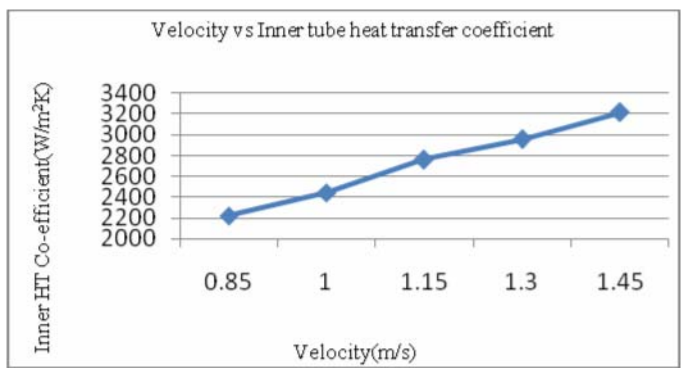

Fig. 5.8 Velocity (m/s) v/s Inner tube heat transfer coefficient (W/m^2K).

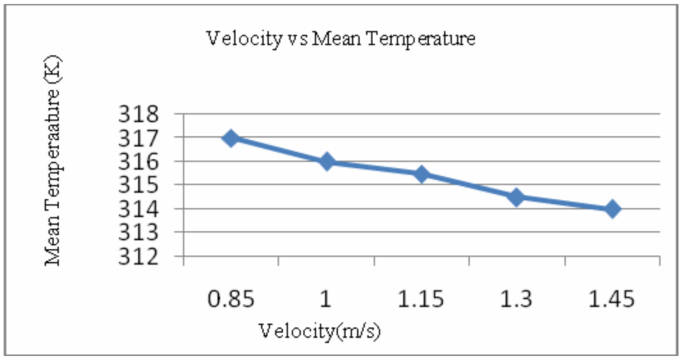

Fig. 5.9 Velocity v/s Mean Temperature

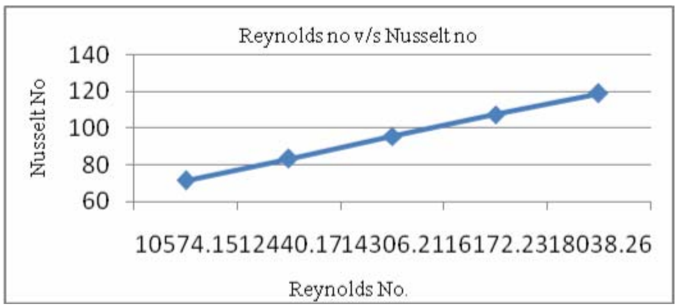

Fig. 5.10 Reynolds no v/s Nusselt no (Nu)

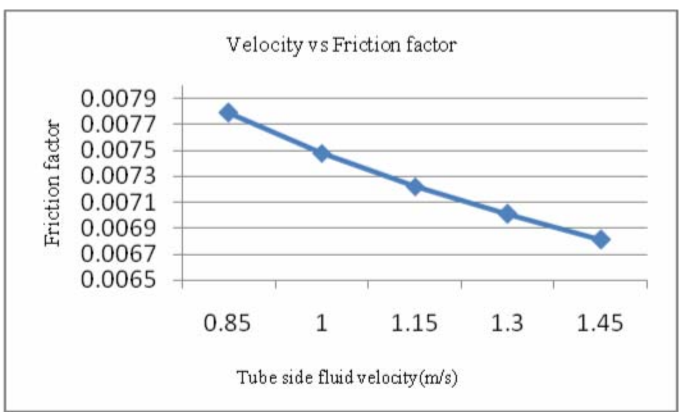

Fig.5.11 Velocity v/s Friction factor

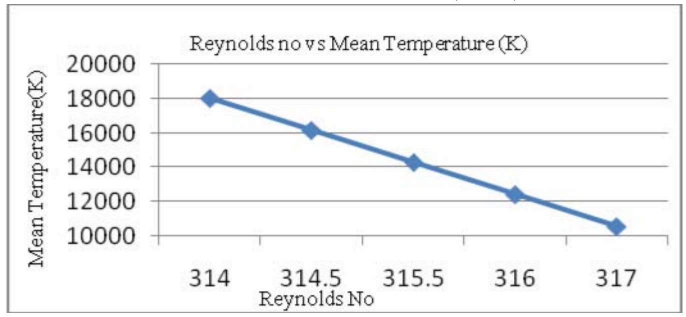

Fig. 5.12 Reynolds no v/s Mean Temperature

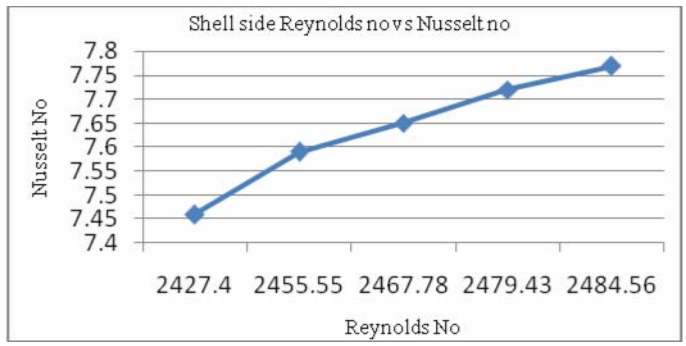

Fig. 5.13 Shell side Nusselt No v/s Reynolds Numbers

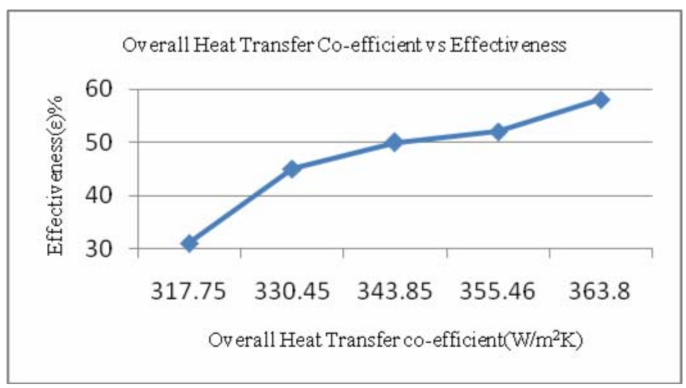

Fig.5.14Overall Heat Transfer Co-efficient vs Effectiveness

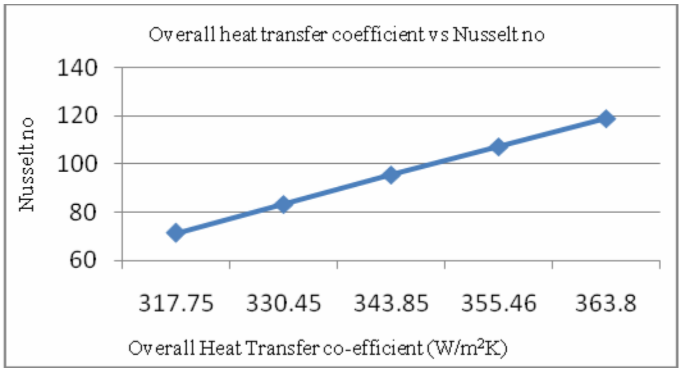

Fig. 5.15 Overall heat transfer coefficient (W/m^2k) v/s Nusselt no

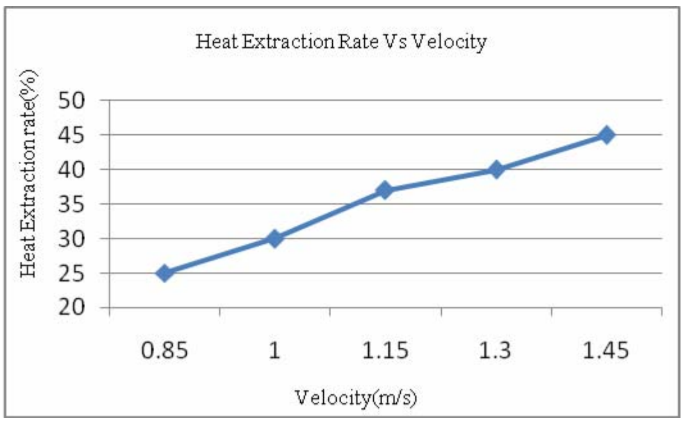

Fig. 5.16 Heat Extraction rate

D. Comparison of the results obtained using CFD with experimental values

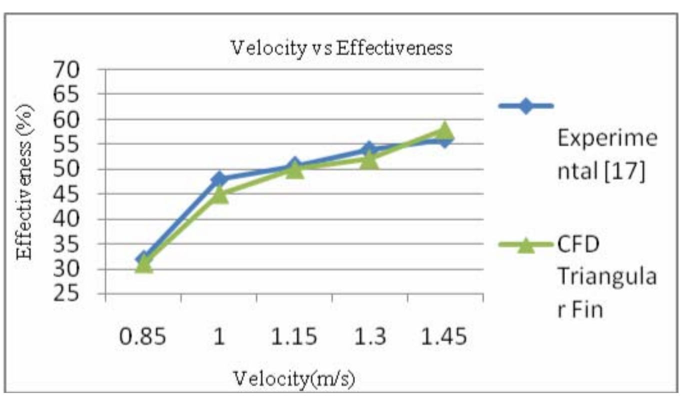

Fig. 5.17 Effectiveness comparison between CFD and experimental values

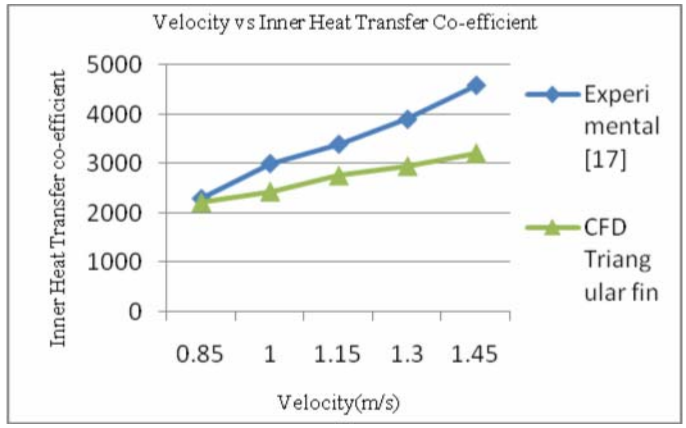

Fig.5.18 Comparison between inner heat transfer coefficient of CFD and experimental values

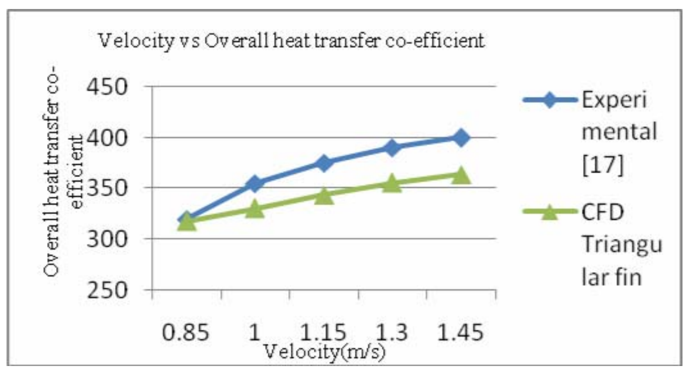

Fig 5.19 Comparison between overall heat transfer co-efficient of CFD and experimental values

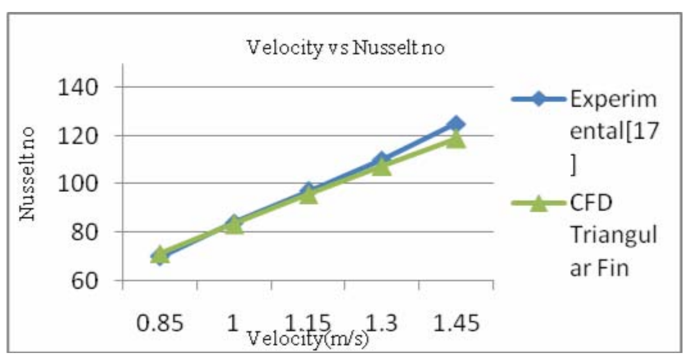

Fig.5.20 Comparison between Nusselt no of CFD and experimental values

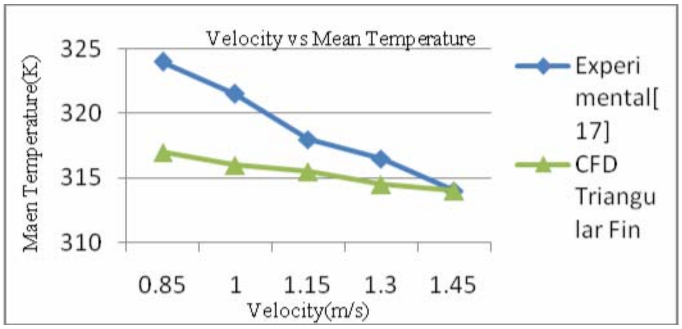

Fig.5.21 Comparison between mean temperature of CFD and experimental values

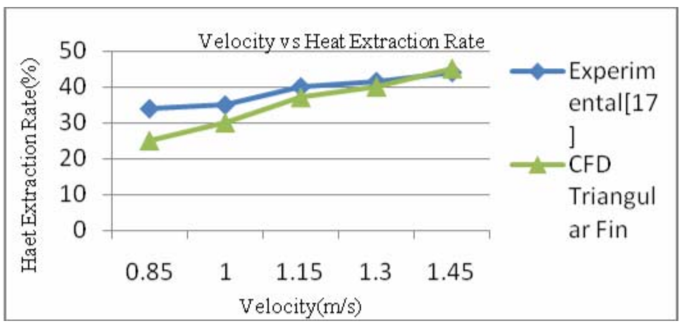

Fig 5.22 Comparison between heat extraction rate of CFD and Experimental values

The comparison between the Experimental effectiveness and CFD effectiveness has been shown in the fig 5.17. It is evident from the plot that the error with both values is in a good agreement. It is given by

$$\text{Perentages of error} = \frac{\text{CFD effectiveness} - \text{Experimental Effectivness}}{\text{CFD effectiveness}}$$

$$\text{Error}(\%) = \frac{50 - 46.8}{50}$$

So Error =6.4 %

The change in this dimensionless number greatly effects the change in the effectiveness of the heat exchanger. The effectiveness also depends on the overall transfer co-efficient (U_0) also in the same way. Friction factor is mainly depended on the tube's inner surface roughness. Due to velocity and viscosity of the fluid effect, it reduces in a small amount though its variation is not that much.

VI. CONCLUSION

An investigation was carried out for predicting the performance of the shell and finned tube heat exchanger computationally. The analysis was completed and the effect on temperature rise and the pressure drop in the finned tube and shell side was observed. Here an approach has been taken to make comparative study of heat transfer between the conventional type square finned tubes and triangular finned tubes. Based on the results obtained from the analysis, the following conclusions have been drawn out.

a) The results from the computational analysis appear to be in good agreement with the available experimental outcomes.

b) The pressure drop in the finned tube is high thereby making the requirement of large pumping power.

c) An alternative to minimize the high pressure requirement is to use shorter length tube.

d) Pressure drop in the shell being very low, there exists no back pressure and hence there is no effect on working in the engine.

e) The effectiveness of the square finned tube heat exchanger is quite analogous with triangular finned tube heat exchanger design.

f) Energy extraction rate is also quite noteworthy which implies that adequate amount of heat can be recovered by the use of triangular finned tube heat exchanger.

g) This Fin arrangement augments maximum heat transfer per unit cost of the equipments than the existing arrangement.

h) The results obtained by triangular fin geometry are quite comparable when taken in conjunction with the fact that the weight and cost factor is greatly minimized with the present geometric fin configuration.

ACKNOWLEDGMENT

At the outset, I would like to take this opportunity to express my sincere gratitude to my mentor Mr. D.H. Das (Associate professor, Mechanical Engineering Department, NIT Silchar) for his valuable guidance, meticulous approach and gracious engagement at each and every stage to make analysis a reality and success.

I am also very much thankful to Mr. AgnimitraBiswas for his help rendered during the whole project work. Finally I would like to thank Mr. Apu Roy, who helped me a lot on the software. I am also deeply grateful to my family members for their unconditional support that gave me the encouragement to complete my M.Tech course.

Last but not the least, heartiest thanks to the Almighty, who bestowed all his blessings on me.

REFERENCES

[1] Simin Wang, Jian Wen , Yanzhong Li, "An experimental investigation of heat transfer enhancement for a shell and tube heat exchanger" *International journal of Applied Thermal Engineering,*Vol- 29 (2009) PP 2433–2438.

[2] R. Hosseini, A. Hosseini-Ghaffar, M. Soltani **, "**Experimental determination of shell side heat transfer coefficient and pressure drop for an oil cooler shell-and-tube heat exchanger with three different tube bundles" *Applied Thermal Engineering,* 27 (2007) 1001–1008.

[3] MansoorSiddique, MajedAlhazmy, "Experimental study of turbulent single-phase flow and heat transfer inside a micro-finned tube", *International journal of refrigeration*, Vol- 31 (2008)P.P 234–241.

[4] Ju-Suk Byuna, JinhoLeeb,, Jun-Young Choi, "Numerical analysis of evaporation performance in a finned-tube heat exchanger", *International Journal of Refrigeration*, 30 (2007) 812-820.

[5] Chi-Chuan Wang a, Wei-Han Tao , Chun-Jung Chang, "An investigation of the airside performance of the slit fin -and-tube heat exchangers", *International Journal of Refrigeration*, 22 (1999) 595-603.

[6] Chi-Chuan Wang, Kuan-Yu Chi, Chun-Jung Chang **, "**A correlation for fin-and-tube heat exchanger having plain fin geometry", *International Journal of Heat and Mass Transfer*, 43 (2000) 2693±2700.

[7] J. Y. Yuna, K.S. Lee. "Investigation of heat transfer characteristics on various kinds of fin and tube heat exchangers with interrupted surfaces ", *International Journal of Heat and Mass Transfer*, 31 "0888# 12641274

[8] Chi-ChuanWanga, , Young-Ming Hwangb, Yur-Tsai Linb "Empirical correlations for heat transfer and flow friction characteristics of herringbone wavy fin-and-tube heat exchangers", *International Journal of Refrigeration*, 25 (2002) 673–680

[9] N. Kayansayan "Heat transfer characterization of plate fin-tube heat exchangers", *Journal of International Refrigeration*, (1994) Vol-17 P-P 117-125.

[10] Mao-Yu Wen, Ching-Yen Ho "Heat-transfer enhancement in fin and tube heat exchanger with improved fin design", *Applied Thermal Engineering*, 29 (2009) 1050–1057 .

[11] C. C. Wang W.L.Fu C. T. Chang "Heat Transfer and Friction Characteristics of Typical Wavy Fin-and-Tube Heat Exchangers", Experimental *Thermal and Fluid Science,* 1997; 14:174-186.

[12] XiaokuiMaa , GuoliangDinga, , YuanmingZhanga , KaijianWangb "The air side heat transfer and friction characteristics of 14 enhanced fin-and-tube heat exchangers with hydrophilic coating under wet conditions", *International Journal of Refrigeration*, 30 (2007) 1153 1167.

[13] Joydeep Barman, A.K. Ghoshal "Performance analysis of finned tube and unbaffled shell-and-tube heat exchangers", *International Journal of Thermal Sciences*, 46 (2007) 1311–1317.

[14] Desai and Bannur, V. Design, "fabrication and testing of heat recovery system from diesel engine exhaust". *Journal of InstEngrs,Vol*- 82,(2001);p.p-111–118.

[15] Morcos VH. "Performance of shell-and-dimpled-tube heat exchangers for waste heat recovery". *Heat Recovery Syst. CHP* 1988;8(4):299–308.

[16] Anderson LZ, Robert Nation H. "Waste heat recovery system for an internal combustion engine". United States patent; 1982 [435-455].

[17] V. Pandiyarajan , M. ChinnaPandian , E. Malan , R. Velraj, R.V. Seeniraj, "Experimental investigation on heat recovery from diesel engine exhaust using finned shell and tube heat exchanger and thermal storage system". International Journal of Applied Energy, 2010:335-350.

[18] http:\\www.en.wikipedia.org\wiki\computational fluid dynamics.

[19] Anderson J.D., *Computational fluid dynamics*, International Edition. New York: McGraw-Hill, 1995.

[20] M. Necatiözisik, Heat Transfer- A basic approach. McGraw-Hill International Editions, pp. 524-566.

[21] Fluent User Guide, Fluent Inc. January 11, 2005.

[22] Amarvirchilka, Ashishkulkarni, modeling turbulent flows in Fluent, product version 6.1.

[23] J.P. Holman, Heat Transfer, McGraw-Hill, edition 2009

[24] C P Kothandaraman, S Subramanyan, Heat and Mass Transfer data book, Fifth edition. New age international publishers, edition 2007

[25] Fuel economy in furnaces and waste heat recovery-PCRA, 2009

AUTHORS PROFILE

S.Sarma, M.Tech, Thermal Engineering, NIT Silchar, Silchar, Assam

D.H.Das, Associate Professor, Dept of mechanical Engineering, NIT Silchar, Silchar, Assam