CFD ANALYSIS FOR SHELL AND TUBE HEAT EXCHANGER FOR WASTE HEAT RECOVERY PROCESS

Nishant Moona, Ajay Kumar Singh

Department of Mechanical Engineering,
Rajiv Gandhi Technical University, Indore, India

ABSTRACT

The high amount of energy is present in the exit or exhaust of many energy conversion devices such as Gas Turbines, I.C. Engines and many more. This energy is totally wasted and is allowed to escape it into the atmosphere. So to avoid this wastage of energy, it should be utilized properly. Thus the energy waste in shell & tube heat exchanger in the form of heat is to be recovered/minimize for obtain maximum efficiency.In present work a Computational Fluid Dynamics (CFD) study was conducted to evaluate the performance of heat exchanger. Some of the key parameters pertaining to heat exchanger such as effectiveness, overall heat transfer co-efficient, energy extraction rate etc. have been reported in this work.

KEYWORDS: *LMTD, Effectiveness, Fouling, NTU, Overall Heat Transfer Coefficient, CFD*

I. INTRODUCTION

Due to the modernization and globalization, the energy resources are depleting very rapidly and very vastly of this world. Thus the price of these precious resources going very high on the international market. So the need to use energy more efficiently has become a necessity. The recovery of waste heat from exhaust gases has become essential due to declining energy resources and production cost. A major result of the energy conversion drive is the development of process recovery aimed at reducing the amount of waste heat discharged to the environment thus increases the overall efficiency of various processes and systems. Heat recovery conserves energy, reduces the overall operating costs and thereby reduces peak loads.

1.1. BACKGROUND OF STUDY

Heat exchangers are devices in which heat is transferred from one fluid to another fluid. Heat exchangers are widely used equipment in various industries such as power generation, refrigeration industry, transportation and process. Heat exchangers are classified on the basis of nature of heat exchange process, relative direction of fluid motion, design and constructional features and physical state of fluids.

a) Nature of Heat Exchange Process:

i. Direct Contact Heat Exchanger: - In this type of heat exchanger, heat transfer takes place by direct mixing of hot and cold fluids. In this process mass exchange also takes place simultaneously.

ii. Indirect Contact Type Heat Exchanger: - In this type of heat exchanger heat is transferred through transmission by wall which separated two fluids.

b) Relative Direction of Fluid Flow:

i. Parallel Flow Heat Exchanger: - In the parallel flow heat exchanger two fluids travels in same direction. Both fluids enter at one end and leaves at the other end.

ii. Counter Flow Heat Exchanger: - In the counter flow heat exchanger hot and cold fluids enter at opposite ends and flow in opposite directions.
iii. Cross Flow Heat Exchanger: - In cross flow two fluids (hot and cold) cross each other in space usually at right angles.

c) Design and Constructional Features
i. Concentric Tube: - Two concentric pipes are used each carrying one of the fluids. The direction of flow may be parallel or counter flow. The effectiveness of heat exchangers is increased by using swirling flow.
ii. Shell and Tube: - In such type of heat exchanger one of the fluids flow through a bundle of tubes enclosed by a shell. The other fluid is forced through the shell and it flows over the outer surface of the tubes.

In addition for transferring the heat for obtaining the basic needs there are certain additional requirements which need to be further specific for the industry in which they are employed, e.g. the exchanger used in automotive and aviation industry need to be light weighted. These exchangers and the exchangers which are used in commercial and domestic refrigeration tends to have the same type of fluid in many applications. The exchangers which are used in chemical process industry are also have a very wide variety of fluid types with different degree of cleanliness. But with the contrast, the exchangers used in cryogenic applications handles relatively clean fluids. These and other similar industry specific requirements have resulted in development of different types of exchanger ranging from the conventional shell and tube heat exchanger to other tubular and non tubular exchangers of varying degree of compactness.

Shell and Tube Heat Exchangers: - The basic principle of operation of these exchangers are very simple and easy, as the two fluids with different temperatures brought into close contact but separated from mixing by some physical barrier. Then the temperature between the two fluids tends to equalize by transfer of heat through the tube wall. This principle is similar to the zeroth law of thermodynamics. The fluids can be either liquids or gases on either the shell or the tube side. In order to transfer heat efficiently, a large heat transfer area should be used, leading to the use of many tubes. In this way, waste heat can be put to use. This is an efficient way to conserve energy.

Shell and tube heat exchangers (STHXs) are widely used in many industrial areas, such as power plants, chemical engineering, petroleum refining, food processing, and etc. A large percentage of world market for heat exchangers is served by the industry workhouse, the shell and tube heat exchanger. According to Master B.I. et al. 2006 more than 35-40% of heat exchangers are of the shell and tube type due to their robust geometry construction, easy maintenance and upgradation. Rugged and safe construction, availability in a wide range of materials, mechanical reliability in service, availability of standards for specifications and designs, and long collective operating experience and familiarity with the designs are some of the reasons for its wide usage in industry. Recent developments in other exchanger geometries have penetrated in various industry applications. Thus the shell and tube heat exchanger still remains the industry choice because of their reliability and maintainability. Over the years, significant research and development efforts were made for the betterment of the shell side geometry. A large variety of different strategies are available for processing and equipment designers to improve industrial heat transfer.

II. OBJECTIVES OF THE WORK

This work gives the analysis of shell and finned tube heat exchanger with the help of CFD in order to see the rise in temperature and pressure drop along the length of the finned tube and the shell. The exhaust gas from the I.C. 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.

Computational Fluid Dynamics (CFD) provides the flexibility to 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.

III. COMPUTATIONAL FLUID DYNAMICS

In a CFD, the most important consideration is how one treats a continuous fluid in a discretized manner on the computer. One method is to discretize the space domain into tiny cells to form a volume mesh or grid mesh and then apply a suitable algorithm to solve the equations of motion i.e. Euler equations for inviscid (non-viscous) and Navier-Stokes equations for viscous flow. In addition into this, such a mesh can be either irregular or regular. So in order to solve directly the Navier-Stokes equations for laminar flow and for turbulent flow when all of the relevant length scales can be resolved by the grid. The other equations are also there which 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, and liquid/solid) or chemically reacting flows i.e. combustion and Non-Newtonian fluids such as blood.

A. Basic Approach to using CFD

i. 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.

ii. Solver:
- Identify and apply conditions at the domain boundary.
- Solve the governing equations on the computational mesh using analysis software.

iii. Post-process: 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

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.

4.1. GEOMETRIC MODELLING

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

Table 1: Heat Exchanger Specifications

1.	Shell Outer Diameter	254 mm
2.	Shell Inner Diameter	240 mm
3.	Shell Thickness	7 mm
4.	Length of the Shell	650 mm

5.	Tube Outer Diameter	18 mm
6.	Tube Inner Diameter	14 mm
7.	Tube Thickness	2 mm
8.	Tube Type	Normal
9.	Type of Tube Layout	Round Layout
10.	Type of Tube Arrangement	In Line Arrangement
11.	No. of Tubes inside the Shell	40

The shell material and tube material is mild steel, and copper respectively. Normal type of tubes has been used and round arrangement has been adopted.

4.2. 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.

4.3. SHELL AND TUBE HEAT EXCHANGER DEPENDS UPON FOLLOWING FACTORS

- Logarithmic Mean Temperature Difference
- Effectiveness
- Fouling
- NTU
- Overall Heat Transfer Coefficient

4.4. CALCULATIONS

a) Heat Transfer:-
Obtained Value: - 11.34 kW

$Q_h = m_h * c_{ph} * [T_{hi} - T_{ho}]$

$Q_c = m_c * c_{pc} * [T_{co} - T_{ci}]$

b) Logarithmic Mean Temperature Difference:- "It is defined as that temperature difference which, if constant, would give the same rate of heat transfer as actually occurs under variable conditions of temperature difference".
Obtained Value: - 20.86 °C

$$Q = UA\Delta\,\theta_m \qquad \theta_m = \frac{\theta_2 - \theta_1}{\ln(\theta_2 / \theta_1)}$$

$$\theta_1 = T_{h,1} - T_{c,1} = T_{h,i} - T_{c,i}$$

$$\theta_2 = T_{h,2} - T_{c,2} = T_{h,o} - T_{c,o}$$

c) Effectiveness: - The heat exchanger effectiveness is defined as the ratio of actual heat transfer to the maximum possible heat transfer.

Obtained Value: - 0.447

$$\varepsilon = \frac{q}{q_{\max}} = \frac{C_h(T_{h,i} - T_{h,o})}{C_{\min}(T_{h,i} - T_{c,i})} = \frac{C_c(T_{c,o} - T_{c,i})}{C_{\min}(T_{h,i} - T_{c,i})}$$

d) Fouling: - In a heat exchanger, during normal operation the tube surface gets, covered by deposits of ash, soot, dirt and scale etc. This phenomenon of rust formation and deposition of fluid impurities is called fouling. Due to these surface deposits the thermal resistance is increased and eventually the performance of heat exchanger low.

Table 2: Fouling Factor for different Fluids

S.No.	Fluid	Fouling Factor (Rf)
1.	Sea Water	0.0001 – 0.0002
2.	Clean river and lake water	0.0002 – 0.0006
3.	Well water	0.0004
4.	Distilled Water	0.0001
5.	Treated boiler feed water	0.0001 – 0.0002
6.	Worst water used in H.E.	<0.0002
7.	Fuel oil and crude oil	0.0009

e) NTU: - NTU stands for Number of Transfer Units which designates the non dimensional heat transfer size of the heat exchanger.
Obtained Value: - 719.07
NTU = U*A/C_{min}

f) Overall Heat Transfer Coefficient: - The overall contains the effect of hot and cold side convection; conduction coefficient is used to analyze heat exchangers.
Obtained Value: - 42.43 W/m^{2o}C

$$\frac{1}{U_i} = \frac{1}{h_i} + \frac{r_i}{k}\ln(r_o / r_i) + (r_i / r_o) * (1/h_o)$$

$$\frac{1}{U_o} = \frac{1}{h_o} + \frac{r_o}{k}\ln(r_o / r_i) + (r_o / r_i) * (1/h_i)$$

$$U = Q / A\theta$$

Table 3: Heat transfer coefficient for fluid combinations

S.No.	Fluid Combination	U(W/m ^{2} ^{o}C)
1.	Water to water	850-1170
2.	Water to oil	110-350
3.	Steam condenser	1000-6000
4.	Alcohol condenser	250-700
5.	Feed water heaters	110-8500
6.	Air condensers	350-780
7.	Air to various gases	60-550

KEY ABBREVIATIONS:-

r_i = inner diameter of tube
r_o = outer diameter of tube
θ_m = LMTD
U = overall heat transfer coefficient
ε = effectiveness
T_{hi} = inlet temperature of hot fluid
T_{ho} = outlet temperature of hot fluid
m_h = mass flow rate of hot fluid
m_c = mass flow rate of cold fluid
c_{ph} = specific heat of hot fluid
c_{pc} = specific heat of cold fluid

V. RESULTS

5.1. RESULT ANALYSIS OF HEAT EXCHANGER BY CFD

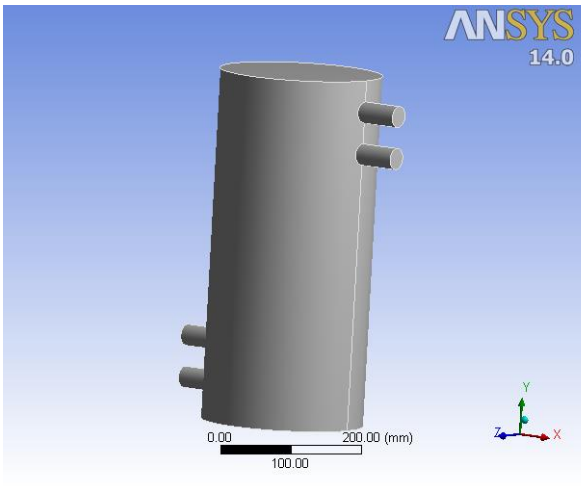

Figure 1: A representation of Shell in 3D

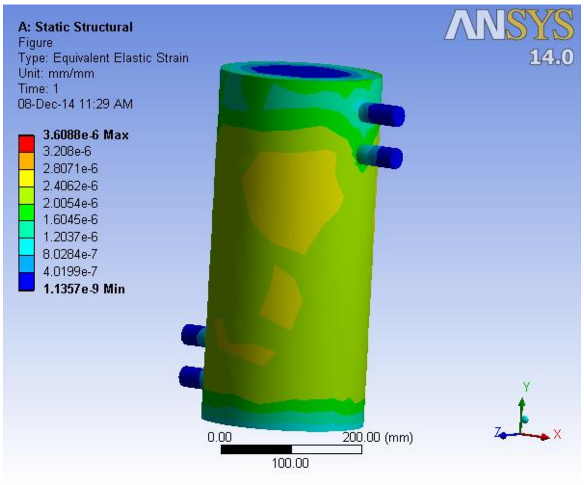

Figure 2: Equivalent amount of Strain in Shell and Tube Heat Exchanger

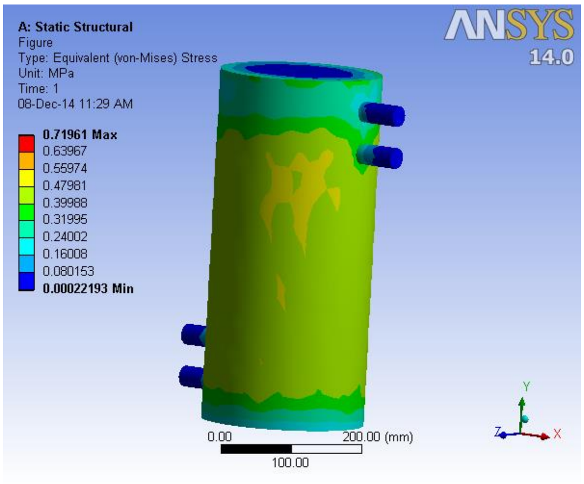

Figure 3: Equivalent amount of Stresses on Shell of Heat Exchanger

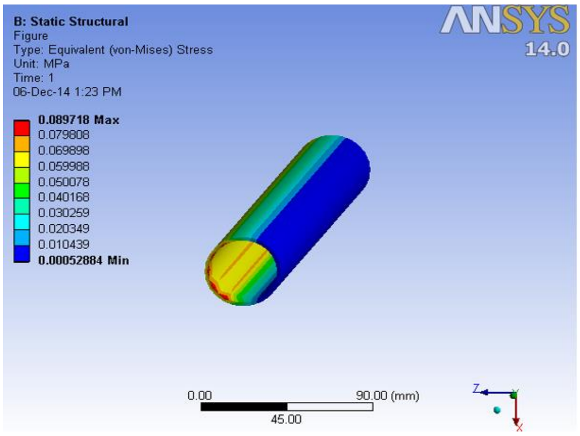

Figure 4: Equivalent amount of Stresses on tube of Heat Exchanger

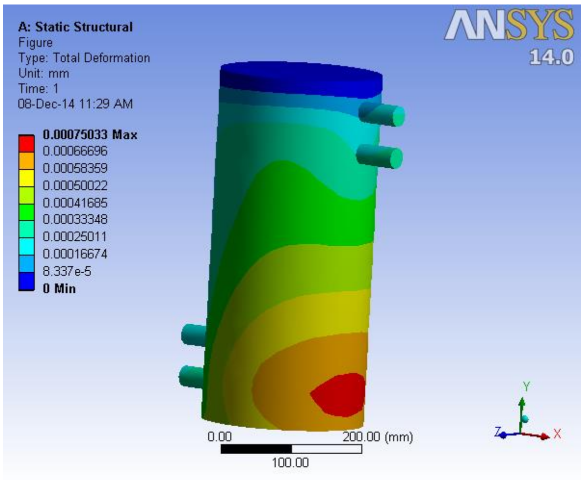

Figure 5: Total amount of deformation on Shell of Heat Exchanger

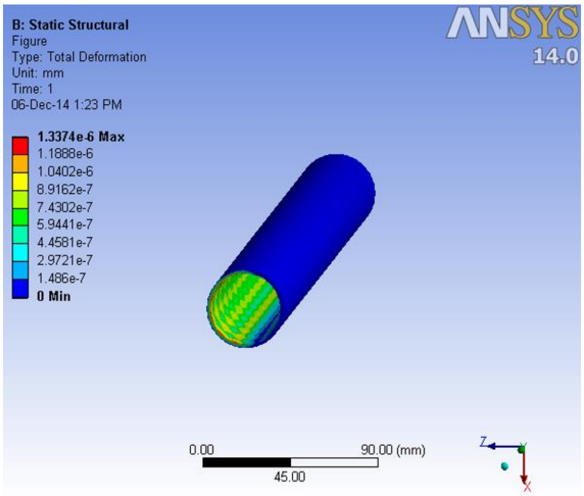

Figure 6: Total amount of deformation on tube of Heat Exchanger

5.2. RESULT ANALYSIS OF HEAT EXCHANGER BY EXPERIMENTALLY

i. Heat transfer rate is 11.34 kW.
ii. Logarithmic Mean Temperature Difference is 20.86 °C
iii. Effectiveness is 0.447
iv. Number of Transfer Units is 719.07

v. Overall heat transfer coefficient is 42.43 W/m^{2o}C

5.3. COMPARING RESULTS

The results which have been taken through experimentally and the results which are to be obtained from CFD are to be compared.

5.4. SUMMARY

The shell and tube heat exchangers are most suitable type of heat exchangers which are widely used in numerous of applications. As they give the best heat transfer rate and has better efficiency in terms of amongst all. The above results are obtained of a shell and tube heat exchanger and further it would be compared to CFD for better heat transfer rate and with due this certain modifications would be achieved.

VI. CONCLUSION

After all calculation related to shell and tube heat exchanger we have measured that the wastage of heat by fouling, frictional pressure drop, effect of environment can be recovered by CFD analysis easily and in less time. According to the data obtained by CFD analysis some implementation in the diameter of tube, size of shell and material of heat exchanger need to be done. After implementation and designing according to CFD we can improve the performance of heat exchanger and increase its efficiency.

VII. FUTURE WORK

If we design the heat exchanger in the future according to CFD analysis then such heat exchanger can improve the heat transfer rate and increase the efficiency. According to this device we can select the fluid which can produce the positive effect on dynamic and thermal parameters like pressure, temperature, velocity, enthalpy etc. We can reduce the negative effect on environment and consumption of power.

REFERENCES

[1] Arjun K.S. & Gopu K.B. (2014) Shell and Tube Heat Exchanger Using Computational Fluid Dynamics Tools, Research General of Engineering Sciences Vol. 3(7).

[2] Gurbir Singh & Hemant Kumar (2014) Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger, Journal of Civil Engineering and Environmental Technology, Vol. 1, No. 3.

[3] Mohammed Reza Saffarian, Tooraj Yousefi & Mostafa Keshavarze Moraveji (2014) Numerical Analysis of Shell and Tube Heat Exchanger with Simple Baffle by CFD, Indian Journal Science Research 7(1)

[4] Sushant Prajapati & M.N. Makwana (2015) CFD Analysis of Shell and Tube Type Heat Exchanger Improvement in Non Continuous Helical Baffle, International Journal for Technological Research in Engineering Vol. 3 Issue 2

[5] Hetal Kotwal & D.S. Patel (2013) CFD Analysis of Shell and Tube Heat Exchanger, International Journal of Engineering Science and Innovative Technology Vol. 2, Issue 2

[6] Alok Vyas & Prashant Sharma (2013) Experiment Analysis Study to Improve Performance of Tubular Heat Exchangers, International Journal of Engineering Research and Applications Vol. 3 Issue 6

[7] Avinash D Jadhav & Tushar A Koli (2014) 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 Vol. 2 Issue 6

[8] Abdur Rahim & S.M. Saad Jameel (2012) Shell side CFD Analysis of a Small Shell and Tube Heat Exchanger considering the effects of Baffle Inclination on Fluid Flow, National Conference on Trends and Advances in Mechanical Engineering, October 19-20, 2012.

[9] Usman Ur Rehman (2011) Heat Exchanger Optimization of Shell and Tube Heat Exchanger through CFD Studies, Department of Chemical and Biological Engineering, Chalmers University of Technology, Sweden.

AUTHORS

Nishant Moona has completed his Engineering in Mechanical Branch in the year 2015 with First Division having Honors and having an aggregate of 80.30%. He was one of the Topper of his Batch. He completed his engineering from Rajiv Gandhi Technical University, Bhopal.

Ajay Singh has completed his Engineering in the Mechanical Branch in the year 2014 with First Division and having an aggregate of 71.80%. He also completed his engineering from Rajiv Gandhi Technical University, Bhopal. Now he is persuing his M.Tech in Rotating Equipments from University of Petroleum and Energy Studies.