ISSN: 0974-2115
www.jchps.com Journal of Chemical and Pharmaceutical Sciences

Model and analysis of shell and tube heat exchanger by using exhaust gases of diesel generator

***Kiran Raj R, M. Sivapragash** and **M. Dev Anand**
Noorul Islam Center for Higher Education, Kumaracoil, Tamilnadu, India.
***Corresponding Author - Email: kiranrajtvm7@gmail.com**

ABSTRACT

In this paper, A 125KVA diesel generator which loses the exhaust gases at a temperature of 350°C. That waste heat is recovered by replacing the silencer of the power plant by a heat exchanger. Heat exchanger is an efficient device in which the process of heating or cooling occurs. The main function of heat exchanger is to provide flow path for hot and cold fluid without mixing of fluids. A host of units know as shell and tube exchangers are built of round tubes mounted in cylindrical shells with there axis parallel to that of the shell. Fluid flow in the Heat exchanger is considered as a fluid dynamic problem and is modelled using finite element method. The flow field in the tube due to the oscillation conditions of inlet flow is analyzed. The pressure pulsation at the tube inlet is employed to produce a flow pulsation, which is caused by superimposing an oscillating pressure gradient on the steady driving pressure of the flow. These are employed as heaters or coolers for a variety of applications. Most heat exchanger problems ultimately result from faulty or inadequate information at the design stage. Computational Fluid Dynamics or CFD is the analysis of system involving the fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation. It is an approach and a tool for analyzing the Fluid Dynamic problems. To implement the CFD, ANSYS software is used as a tool. Now a day's fluid flow analysis is a challenging criteria and is even in research to obtain the flow linearity and complete fluid flow at various environmental conditions.

KEYWORDS: Computational Fluid Dynamics, Heat Exchanger, Fluid Flow, ANSYS

1. INTRODUCTION

In a 125KVA diesel generator the exhaust gases (flue gases) are leaving the chimney at a temperature of about 350°C presently the exhaust gases are waste to the atmosphere. This waste heat is recovered by using a heat exchanger placed instead of silencer of the power plant. The flue gases are cooled using a shell and tube heat exchanger. A wide variety of heat exchangers are employed in industrial applications which is used to exchanger heat between two fluids either in direct contact (or) indirect contact with each other (Hatami, 2015). Exchangers have variety of names depending on their modes of heat transfer. Heat flow in the tube has received significant attention in thermal engineering due the enhancement of heat transfer coefficients. In general, the pulsating flow field consists of a steady flow part and as oscillating pressure (Vamsi, 2015). In this study we have considered only pressure distribution of the pulsating flow. A wide variety of heat exchangers are employed in industrial applications, which can be served for this equipment is to exchange heat between two fluid either in direct contact or indirect contact with each other. Exchanger has variety of names depending on their modes of heat transfer. These names serve to identify the modes of heat transfer of the system and in many cases also infer the image of the shape and appearance of the unit Rating Engineering recognize that heating partial or total Condensation highly viscous flows etc. are considerations, which may profoundly affect the desirable shape of the heat exchanger.

Shell and tube heat exchanger: As a piece of mechanical hardware, a tubular heat exchanger consists of two Intertwined pressure vessels. The inlet header, outlet header, inside of the tubes and the Inlet / outlet nozzles commonly referred to as the "tube side" chamber. The remaining space in the heat exchanger between the shell and the tube is the other pressure vessels Known as the "shell side" chamber. Two fluid at different temperatures enter the two pressure chambers exchange across the tube walls throw a combined conductions- convection mechanism and then exit throw the outlet nozzle. The shell and tube heat exchanger is the most common of the various types of unfired heat transfer equipment used in industry. It is recuperative type heat exchanger. Although it is not especially compact, it is robust and it can be designed for large surface areas having pressure greater than 30bar and temperature greater than 260°C (4-18).

Design parameters:

C Hot Fluid	: Flue Gases
Cold Fluid	: Water

Specification

Shell Material	= Mild Steel
Tube Material	= Stainless Steel
Thi	= 3500C
Tci	= 300C
Tho	= 1000C
Tco	= 1600C

Assume tube OD = 19.05mm
Thickness of tube = 1.24446mm
Tube ID = 16.56mm
Shell OD = 334mm
Shell ID = 322mm
Length of the Exchanger L = 1.8m
Volume flow Rate of Flue Gas from the Diesel Engine = 10,000m^3/Hr
Flue Gas Velocity = 95m/sec
Quantity of Fuel Consumption per Hour of Diesel Engine = 126 Lit/Hr
To Find the Mass Flow Rate of Flue Gas
∴ Q = AV = 10,000 m^3/hr

$$\frac{m}{p} = \frac{10,000}{3600}$$

$$\therefore m_f = \frac{10,000\ X\ 0.571}{3600}$$

$$m_f = 1.586\ kg/sec$$

This Mass Flow Rate can be Divided into Two Silencers.
∴ Mass Flow rate Per Silencer

$$m_f = \frac{1.586}{2} = 0.793\ kg/sec$$

Properties of Fluids:
Hot Fluid (Flue Gas)

ρ_h = 0.571 kg/m^3
C_{Ph} = 1.1365 KJ/Kgk
M_h = 29.95 X 10^{-6} kg/ms
P_r = 0.645
K_h = 52.685 X 10^{-3} w/ mk

Cold Fluid (Air)

ρ_c = 0.959 kg/m^3
C_{Pc} = 1.165 kg / kgk
m_c = 21.675 X 10^{-6} kg/ms
P_r = 0.689
K_c = 31.69 X 10-3 w / mk

Mass Flow Rate of Cold Fluid

Q = $m_h\ C_{Ph}\ (\Delta T)_h$
= $m_c\ C_{Pc}\ (\Delta T)_c$
0.793 X 1.1365 X (350 – 100) = mc X 1.165 X (160 – 30)
∴ mc = 1.4877 kg / sec

Capacity Rating of the Blower

Q = Ava

$$\frac{m_c}{\rho} = \frac{1.4877}{1.165}$$

Q = 1.2769 m^3 / sec
1m^3 = 35.33 ft^3

$$Q = \frac{1.2769\ X\ 35.33}{\frac{1}{60}} = \quad = 2700\ C.\ F.\ M$$

∴ The Capacity Rating of the Blower = 2700 C. F. M

Total Area of Discharge Pipe of Blower

A = 254 X 254
A = 0.0645 m^2

Velocity of Air From the Blower

Q = AV_a
1.2769 = 0.0645 X Va
∴ Va = 19.79 m / sec

Actual heat transfer surface area:

Q = FUA LMTD
Where F – Correction Factor

U – Overall Heat Transfer Coefficient 250 (Assumed Value)
LMTD – Log Mean Temp Difference
A – Surface Area
Assume : flow in Counter Direction LMTD for Counter Flow

$$LMTD = \frac{(\Delta T_1)-(\Delta T_2)}{In\,(\frac{\Delta T_1}{\Delta T_2})}$$

$$= \frac{(350-160)-(100-30)}{In\,(\frac{350-160}{100-30})}$$

LMTD = 120.18^{0}C

Correction factor: The correction factor is a function of the shell and tube fluid temperatures and the number of tube and shell pass. It is normally correlated as a function of two dimension less temperature ration.

$$R = \frac{T_{ci}-T_{co}}{T_{ho}-T_{hi}} = \frac{30-160}{100-350} = 0.52$$

$$P = \frac{T_{ho}-T_{hi}}{T_{ci}-T_{co}} = \frac{100-350}{30-160} = 1.923$$

From HMT Data Book Page No. 161

F= 1

Stream flow rates: Allocating the fluids with the lowest flow rate to the shell inside normally give the most economical design.

Q = FUA LMTD

$$A = \frac{225.3\ X\ 10^3}{1\ X\ 250\ X120.18}$$

To find the no" of tubes

A = nπdo L

7.499 = n X π X 19.05 X 10-3 X 1.8

∴ n = 69.61 ≏ 70

n = 70 tubes

Equivalent Diameter

For square pitch

$$de = \frac{4(pt^2-\pi d^2/4)}{\pi d_0}$$

$$de = \frac{1.27}{d_0}\,(pt^2 - 0.785{d_0}^2)$$

Where Pt = 1.25 d_o

= 1.25 (19.05)

P_t = 23.8125 mm

$d_e = \frac{1.27}{0.01905}(0.0238125^2 - 0.785 \times 0.1905^2)$

d_e = 0.01881 m

Tube Sheet Lay – Out

Number of Passes	1	2	4	6	8
K1	0.215	0.156	0.158	0.0402	0.0331
n1	2.207	2.291	2.263	2.617	2.643

The bundle diameter will depend not only on the number of tubes but also on the number of passes, as spaces must be left in the pattern of tubes on the sheet to accommodate the passes partition. As an estimate of the bundle Db can be obtained from the following equation.

$N_t = k_l\,(D_b / d_o)^n 1$

$D_b = d_o\,(N_t / k_l)^{l/n} 1$

Where

N_t = Number of Tubes

D_b = Bundle Diameter in mm

D_o = Tube Outside Diameter in mm

Constant for Use

Square Pitch P_t = 1.25 d_o

For Single Pass Arrangement

K_1 = 0.215

n_1 = 2.207

∴ $D_b = d_o\,(N_t/K_l)^{1/n}1 = 19.05\,(70 / 0.215)^{1/2\ 207}$

D_b = 262 mm

Allowance = 18.71 mm

∴ Bundle Dia, D_b = 262 + 18.71= 280.76 mm

Shell Inner Diameter = 280.76 + 25.4= 306.16 mm

Shell Outer Diameter = 306.16 + 25.4= 331.56 mm

Shell O. D = 331.56 mm < 334 mm

So the Design is Safe

Baffle Spacing

Baffle Diameter = D_s – 1 / 16^{in}= 322 – 1.6= 320.4 mm

Baffle Spacing l_b = 0.4 X D_s= 0.4 X 322= 128.8 mm

Cross Flow Area Calculation:

$$A_s = \frac{(P_t-d_0)D_s\ X\ l_b}{P_t} = \frac{(23.8125-19.05) X\ 322\ X\ 128.8}{23.8125}$$

A_s = 8294 mm^2

A_s = 0.008294 m^2

Shell side mass velocity

$$Gs = \frac{m_s}{A_s} = \frac{0.793}{0.008294} = 95.6 \text{kg/ m}^2\text{s}$$

Reynolds Number Calculation

$$Re = \frac{G_s d_e}{\mu} = \frac{95.6\ X\ 0.01881}{29.95\ X\ 10^{-6}} = 6,0041$$

Re = 60041 > 10,000

So, the Flow becomes Turbulent

$$Nu = 0.36\ (Re)^{0.55}\ (Pr)^{0.33} \left(\frac{\mu}{\mu_w}\right)^{0.14} = 0.36\ (60041)^{0.55}\ (0.645)^{0.23}\ X\ 1$$

$$\frac{h_o d_e}{k_s} = 132.31$$

$$h_o = \frac{132.31\ X\ 52.685\ X\ 10^{-3}}{0.01881}$$

h_o = 370.6 w/m^2 k

Tube Side Parameters

$$Re = \frac{p\ v d_i}{\mu} = \frac{0.959\ X\ 19.79\ X\ 16.56\ X\ 10^{-3}}{21.675\ X\ 10^{-6}}$$

Re = 14500 > 10,000

∴ So the flow becomes turbulent

Nu = 0.023 $(Re)^{0.8}$ $(Pr)^n$= 0.023 $(14500)^{0.8}$ $(0.689)^{0.4}$

$$\frac{h_i d_i}{k_t} = 42.28$$

$$\therefore h_i = \frac{42.28\ X\ 31.69\ X\ 10^{-3}}{16.56\ X\ 10^{-3}}$$

$$h_i = 80.90\ W/m^2k$$

To Find the Overall Heat Transfer Coefficient Neglecting the Fouling Resistances

$$\frac{1}{U_0} = \frac{1}{h_0} + \frac{1}{h_i}\left(\frac{d_0}{d_i}\right) = \frac{1}{370.6} + \frac{1}{80.9}\left(\frac{19.05}{16.56}\right)$$

$$U_0 = 59.1\ w\ /m^{20}\ k$$

The value obtained is which in the limit of value assumed so the design is safe.

Pressure Drop Calculations

Tube Side Pressure Drop

$$\Delta P_t = \frac{f\ Gt^{2\ Ln}}{2\ \rho d_i\ \emptyset_t}$$

f = 0.079 $Re^{-0.25}$= 0.079 $(14500)^{-0.25}$

f = 0.007199

$$\Delta P_t = \frac{0.007199\ X\ (98.65)^2\ X\ 1.8\ X\ 1}{2\ X\ 0.959\ X\ 16.56\ X\ 10^{-3}\ X\ 1}$$

ΔP_t = 3970 N / m^2

ΔP_t = 3.97 kN / m^2 < 10kN / m^2

So, the Design is Safe

Shell Side Pressure Drop

$$\Delta P_s = \frac{f\ Gs^2 D\ S^L}{2\ X\ 10^6\ X\ d_e\ X\ l_s}\ KN\ /\ m^2$$

f = 1.78 $Re^{-0.2}$ = 1.78 $(60041)^{-0.2}$

f = 0.1971

$$\therefore \Delta P_s = \frac{0.1971\ X\ (95.6)^2\ X\ 0.322\ X\ 1.8}{2\ X\ 10^6\ X 0.01881 X\ 0.1288} = 0.2155 \text{kN / m}^2 < 10 \text{kN / m}^2$$

ΔP_s = 215.5 N / m^2

The pressure drops are within the limits so the design is safe

Modelling of heat exchanger using CREO 2.0: From the above calculations the shell and tube exchanger deign in CREO 2.0 and the model is shown below Fig. 1 and 2 CREO 2.0 is parametric solid modeling system - models are defined by dimensions which are easy to change and the models have some 'intelligence'. CREO 2.0 is a good way of implementing concurrent engineering, which is the way design is increasingly organized. 'Concurrent engineering is a systematic approach to the integrated, concurrent design of products and their related processes, including manufacturing and support'.

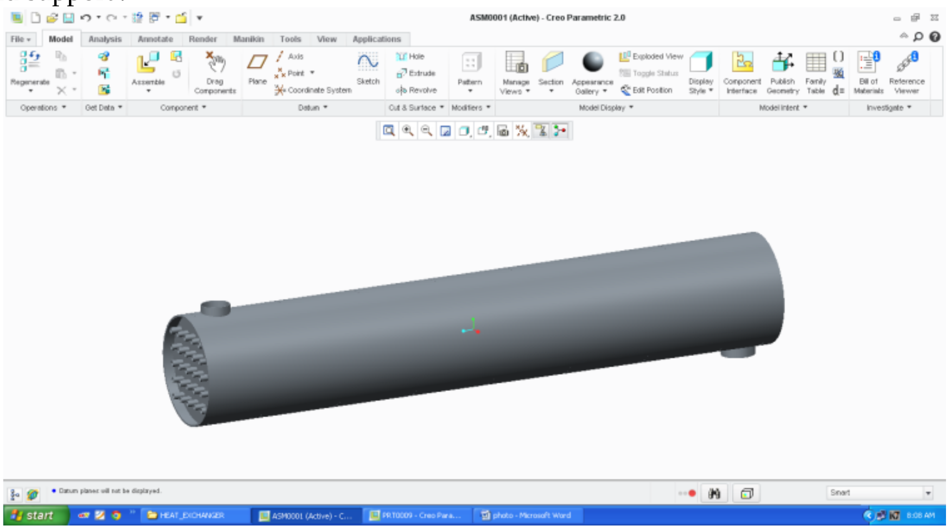

Fig.1.Solid Modeling System

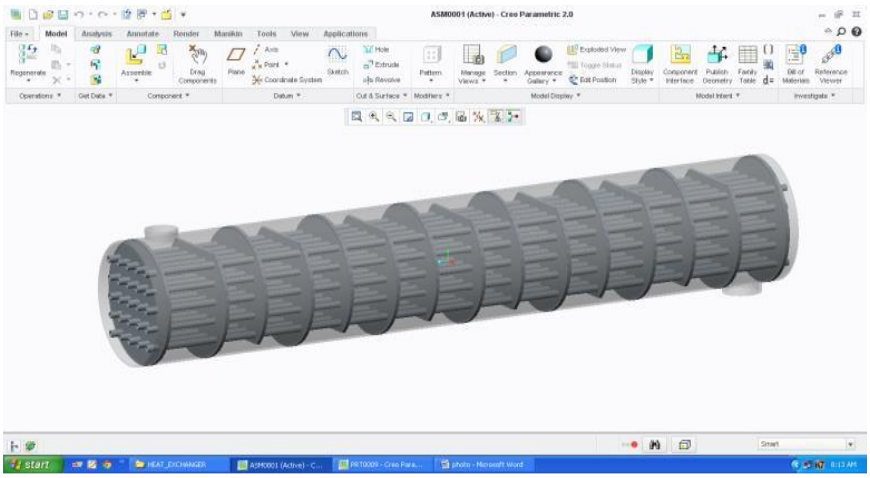

Fig.2.Grid Modeling System

SIMULATION

Modal Analysis: Modal analysis is conducted to find thermal analysis of shell and tube heat exchanger.

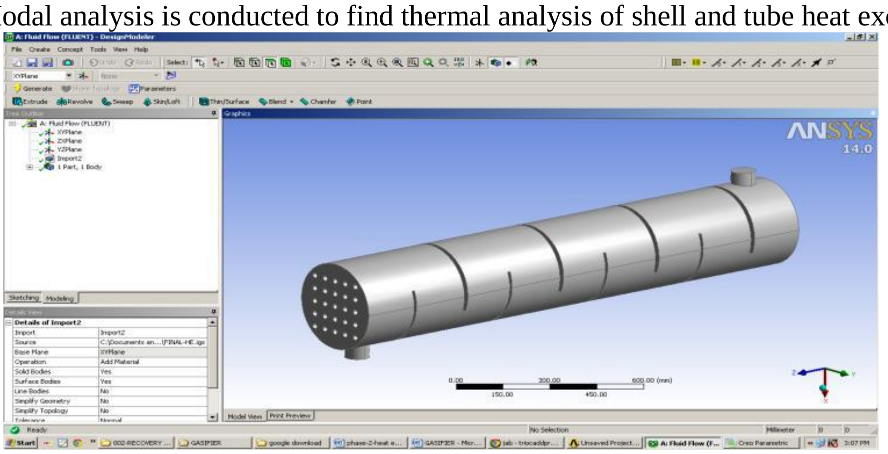

Fig.3.Solid Modeling System Analysis

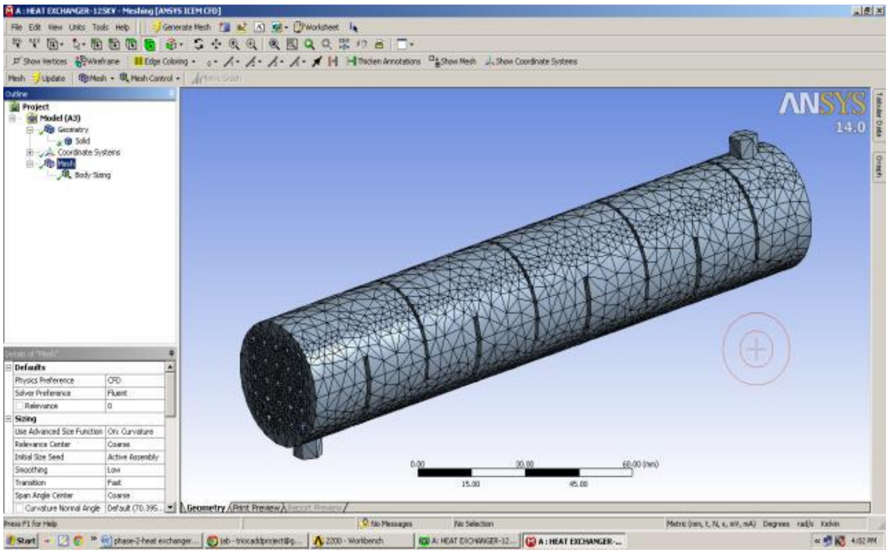

Fig.4.Grid Modeling System Analysis

CFD analysis: Simulation is conducted to find out the advanced geometry acquisition, mesh generation, mesh optimization, and post-processing tools to meet the requirement for integrated mesh generation and post processing tools for today's sophisticated analyses.

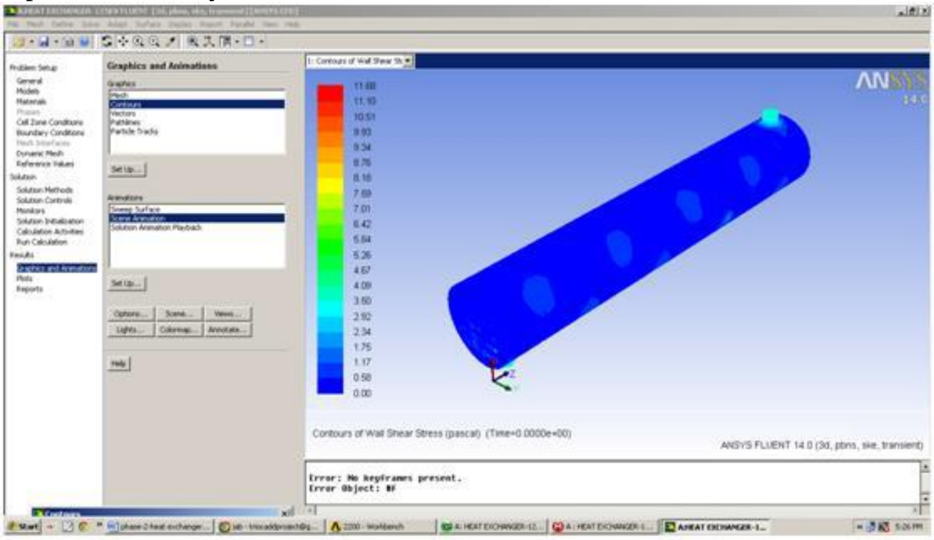

Fig.5.Integrated Mesh Generation

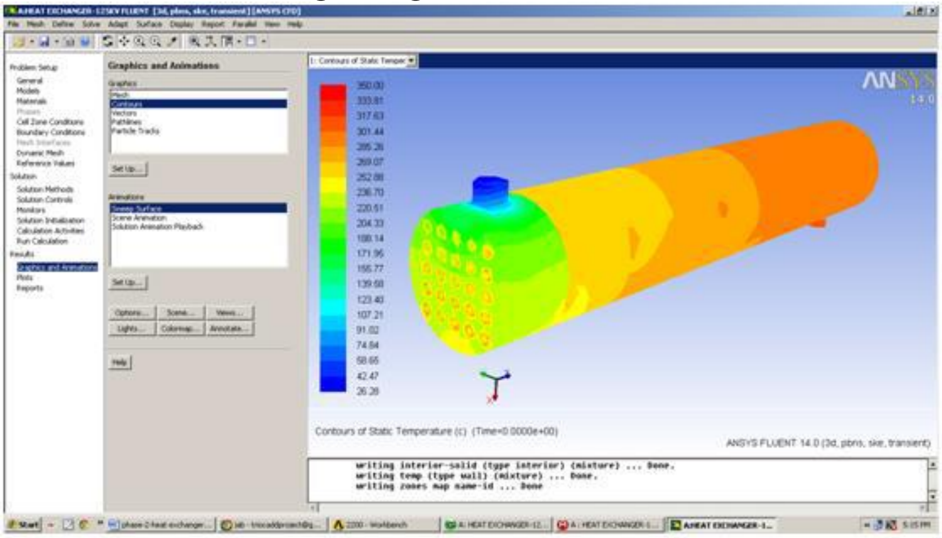

Fig.6.Post Processing

2. CONCLUSION

Conventional design does not give real heat transfer effect. To overcome such problems step is taken for a new design of a counter flow shell and tube type heat exchanger. Heat exchanger dimensions are calculated initially by design procedure. Then the calculations are used for modeling. Thus the modeling is created successfully in CREO 2.0. While carrying out this project we are able to study about the 3D modelling software (CERO 2.0) to develop our basic knowledge to know about the industrial design. The modeling is done by the above procedure. Next the modeled Heat exchanger is to be Computational Fluid Dynamics (CFD) software ANSYS FLUENT. From this work we conclude about 72% of heat is recovered and 28% heat is lost to the atmospheres.

REFERENCES

Andre L.H. Costa, Eduardo M. Queiroz, Design Optimization of Shell and Tube Heat Exchangers", Applied Thermal Engineering, 28, 2008, 1798–1805.

Babu BV, Munawarb SA, Differential Evolution Strategies for Optimal Design of Shell and Tube Heat Exchanger, Chemical Engineering Science, 62, 2007, 3720 – 3739.

Bari S, Hossain SN, Design and Optimization of Compact Heat Exchangers to be Retrofitted into a Vehicle for Heat Recovery from a Diesel Engine, 6th BSME International Conference on Thermal Engineering (ICTE 2014), 2014.

Fesanghary M, Damangir E, Soleimani, Design Optimization of Shell and Tube Heat Exchanger Using Global Sensitivity Analysis and Harmony Search Algorithm, Applied Thermal Engineering, 29, 2009, 1026–1031.

Hatami M, Ganji DD, Gorji-Bandpy M, Experimental and Thermo-dynamical Analyses of the Diesel Exhaust Vortex Generator Heat Exchanger for Optimizing Its Operating Condition" Babol University of Technology, Mechanical Engineering Department, Babol, Iran, 2015.

Hossain SN, Bari S, Effect of Different Working Fluids on Shell and Tube Heat Exchanger to Recover Heat From Exhaust of an Automotive Diesel Engine, Sustainable Energy Centre, School of Advanced Manufacturing and Mechanical Engineering, University of South Australia, SA 5095, Australia

Jiangfeng Guo, Lin Cheng, Mingtian Xu, Optimization Design of Shell and Tube Heat Exchanger by Entropy Generation Minimization and Genetic Algorithm, Applied Thermal Engineering, 29, 2009, 2954–2960.

Khushnood S, Cross-Flow-Induced-Vibrations in Heat Exchanger Tube Bundles: A Review‟, University of Engineering & Technology, Taxicab Pakistan, 1993.

Patel VK, Rao RV, Design Optimization of Shell and Tube Heat Exchanger Using Particle Swarm Optimization Technique, Applied Thermal Engineering, 30, 2010, 1417-1425.

Resat Selbas, Onder Kızılkan, Marcus Reppich, A New Design Approach for Shell and Tube Heat Exchanger Using Genetic Algorithms From Economic Point of View, Chemical Engineering and Processing, 45, 2006, 268–275.

Ryu KW, Park CY, Lee H, Effects of Support Structure Changes on Flow-Induced Vibration Characteristics of Steam Generator Tubes, Nuclear Engineering Technology, 42, 2010, 97–108

Saiful Bari, Shekh N Hossain, Waste Heat Recovery from a Diesel Engine Using Shell and Tube Heat Exchanger, Barbara Hardy Institute, School of Engineering, University of South Australia, Mawson Lakes Campus, SA 5095, Australia, 2013.

Sandeep K. Patel, Shell & Tube Heat Exchanger Thermal Design with Optimization of Mass Flow Rate and Baffle Spacing, International Journal of Advanced Engineering Research and Studies, 2(1), 2012, 130-13.

Sandeep R. Desai, S. Pavitran, Determination of Natural Frequency of A Normal Square Tube Array, International Journal of Engineering Sciences Research , July 2011, 67-71

Sepehr Sanaye, Hassan Hajabdollahi, 2010, "Multi-Objective Optimization of Shell and Tube Heat Exchanger", Applied Thermal Engineering, 30, 1937-1945

Shravan H Gawande, Appasaheb A Keste, Laxman G. Navale, Milindkumar R. Nandgaonkar, Vaishali J. Sonawane, Umesh B. Ubarhande, Design Optimization of Shell and Tube Heat Exchanger by Vibration Analysis‟, Modern Mechanical Engineering, 1, 2011, 6-11.

Vamsi Mokkapati, Chuen-Sen Lin, Numerical Study of an Exhaust Heat Recovery System Using Corrugated Tube Heat Exchanger with Twisted Tape Inserts, Mechanical Engineering Department, University of Alaska Fairbanks, P.O. Box 755905, Fairbanks, AK, 99775-5905, USA, 2015.