Chapter 3

Performance Analysis of Smoke Tube Boiler for Waste Heat Recovery

Shaikh Alam[1], Shriram M. Ughade[2*]

[1]*Department of Mechanical Engineering, RKNEC Nagpur (MH), India*
[2]*Department of Mechanical Engineering, SCET Nagpur (MH), India*

ABSTRACT

Waste heat recovery is the recovery of waste heat from hot fluids that can be reused again for other processes. Waste heat recovery can be done through various waste heat recovery technologies to provide valuable energy sources and reduce the overall energy consumption. In the present work, performance analysis of smoke tube boiler for waste heat recovery is carried out. The main objectives of this work are to analyze the existing waste heat recovery system for smoke tube boiler and carryout simulation of existing waste heat& improvement in waste heat recovery system for smoke tube boiler. For validation, analytical results are compared with simulation results.

Keywords: Waste heat recovery, Boiler, heat exchanger, mixing chamber, mass transfer

1. Introduction

Industrial waste heat refers to energy that is generated in industrial processes without being put to practical use. Sources of waste heat include hot combustion gases discharged to the atmosphere, heated products exiting industrial processes, and heat transfer from hot equipment surfaces. Waste heat recovery entails capturing and reusing the waste heat in industrial processes for heating or for generating mechanical or electrical work. Example uses for waste heat include generating electricity, preheating combustion air, preheating furnace loads, absorption cooling, and space heating. Heat recovery technologies frequently reduce the operating costs for facilities by increasing their energy productivity. Many recovery technologies are already well developed and technically proven; however, there are numerous applications where heat is not recovered due to a combination of market and technical barriers [14].

In the present work, performance analysis of smoke tube boiler for waste heat recovery is carried out at JSW Steel Coated Products Limited Kalmeshwar, Nagpur (MH). In the existing waste heat recovery smoke tube boiler, volatile organic compound from prime oven and finish oven of color coating line is sucked by exhaust blower and passes through Regerative Thermal Oxidizer (RTO) where volatile organic compound gets decomposes into flue gas. The flue gas is passed in mixing chamber where the temperature of flue gases reduces by mixing of proper amount of fresh air supplied by blower in mixing chamber. The flue gases is then passed in heat exchanger where flue gases and fresh air is heated. The temperature of outlet fresh air after heat exchanger is maintained by controlling the bypass valve which control the air flow to chimney. The heat recovered from heat exchanger is then send to the prime and finish oven of color coating line for preheating process. The heat exchanger then supplies the flue gases to boiler where temperature of gases gets reduced and this gases is passed in atmosphere through chimney.

Chapter 3

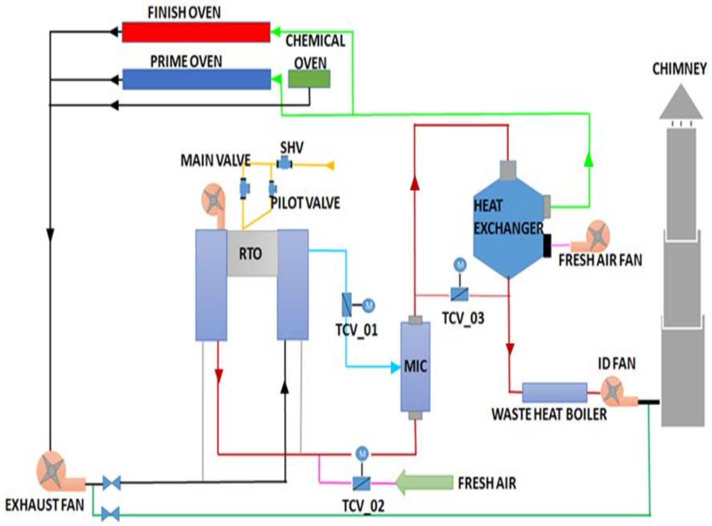

Fig.1. Existing waste heat recovery smoke tube boiler system

Nomenclature

Q	Heat transfer rate (w)
L	Tube length (m)
t	Tube thickness (m)
Nt	No. of Tubes (units)
Cp	Specific Heat Capacity (kJ/kgK)
Thi	Hot fluid inlet Temperature (K)
Tci	Cold fluid inlet Temperature (K)
Tho	Hot fluid outlet Temperature (K)
Tco	Cold fluid outlet Temperature (K)
Uo	Overall heat transfer coefficient (W/m²k)
Di	Tube inside diameter (m)
Do	Tube outside diameter (m)
De	Equivalent diameter (m)
Ds	Diameter of shell (m)
Nu	Nusselt number
Pr	Prandtl number
Re	Reynolds number
Gs	mass flux (kg/m²s)
µ	kinematic viscosity (m²/s)
ρ	Density (kg/m³)
Kmt	Thermal conductivity of tube material (W/mK)
hi	Shell side heat transfer coefficient (W/m²k)
ho	Tube side heat transfer coefficient (W/m²k)
ΔPs	Shell side pressure drop (N/m²)
ΔPt	Tube side pressure drop (N/m²)
ΔTm	log mean temperature difference(℃)

Abbreviations

CTP	tube count calculations constant
CL	tube layout constant
STHE	Shell and tube heat exchanger

LMTD	Logarithmic mean temperature difference
PR	Pitch ratio
PT	Tube pitch
t	tube side flow

Subscript

c	cold fluid
h	hot fluid
i	inlet conditions
o	outlet conditions
s	shell side flow

2. LITERATURE REVIEW

Panagiotis Drosatos et al. proposed an in-house built code and its incorporation into an ANSYS FLUNT Computational Fluid Dynamics (CFD) model for the simulation of the convection section of a boiler.

A. Zargoushi et al. Developed a CFD model in ANSYS FLUENT to understand transport phenomena, especially the phase change in a complex plate-fin heat exchanger operated in a gas refining company. Three models local thermal equilibrium between the porous medium and fluid flow without mass transfer(LTE-non mass), local thermal equilibrium between the porous medium and fluid flow with mass transfer(LTE), and local thermal non- equilibrium between the porous medium and fluid flow with mass transfer(LTNE) have been evaluated in this paper. Among three model, LTNE with mass transfer provides better results than those of LTE and LTE without mass transfer.

Chamil Abeykoon et al. designed and carried out CFD simulation of compact heat exchanger. In this work, six CFD models were developed. On the basis of CFD results, it was found out that a selection of parameters such as a the baffle cut ratio, number of baffles , tubes flow and tube arrangements are important in optimizing the performance of a shell and tube heat exchanger.

Wei-Wei Wang et al. carried out the mathematical and CFD numerical investigation of a radial heat pipe.CFD results indicates that present built volume of fluid model can effectively reproduce phase change fluid flow and heat transfer inside radial heat pipe. Numerical modeling results were validated by the experimental tests for various conditions and maximum deviation fall within 10%.

M.A. Gomez et al. studied about the influence of exhaust gas recirculation and oxygen excess ratio in a biomass fed with pure oxygen. A theoretical model for simulation of biomass combustion is proposed through additional in built routines on a CFD commercial code. The model implements several sub models of thermal conversion of solid biomass as well as heat and mass transfer, chemical reactions and radiation transport to solve the behavior of a boiler in a steady situation with a relatively low computational cost.

Ram Thakar et al. Recovered waste heat energy of exhaust gas of diesel engine by placing specially designed heat exchanger near the inlet and outlet duct of the engine in order to used exhaust energy for preheating the air passed to the engine. The heating of inlet air improves the performance of the diesel engine and reduces the emission level. The effectiveness of heat exchanger found to be 0.615.

Elloit Woolley et al. studied how an industrial waste heat recovery is done. They studied about availability of waste heat energy, the ability to recover and to reduce industrial energy costs and environmental impacts. They developed a waste heat energy recovery framework to provide manufacturers with a four step methodology in assessing production activities in facilities, analyzing the compatibility of waste heat source sink in term of energy balance and temporal availability, selecting appropriate heat recovery technologies and decision support based on economic benefits.

M.C. Barma et al. studied about amount of energy used in boiler, various losses and causes occurred in boilers and how to minimized heat loss using various technologies. Energy saving measures in boiler is also discussed in this study. Different types of energy saving techniques such as excess air control, improving combustion efficiency, utilization of waste heat content of flue gases etc., have been reviewed in this study.

Ratchaphon Suntivarakorn et al. carried out the experiment to improve the efficiency of a fire tube boiler with a fixed gate and screw conveyor for feeding fuel. The experiment result shows that using heat recovery and fuel drying reduces 3% weight of fuel moisture content and boiler efficiency increases 0.41%.

Shubham Agarwal et al. studied about boiler maintenance and various possible causes responsible for the breakdown of the boiler. Different research approaches have been discussed related to the material of the boiler and its components, regarding various hazards possible in the boilers. A maintenance schedule has also been prepared so as to optimize different maintenance actions.

M. Manickam et al. developed model of waste heat boiler for utilizing plant off-gas consisting of gaseous and particulate combustibles. This model can allows various calculations like temperatures of gas and particles within the boiler. Mixing in the burner region, char burnout and char particles temperature were analyzed using this model. Combustion stability also studied using Eddy break-up model which accounts for combustion kinetics and the results obtained are compared with a mixed-is-burnt model.

3. OBJECTIVES

a) To analyze the existing waste heat recovery system for smoke tube boiler.
b) To carryout simulation of existing waste heat recovery system for smoke tube boiler.
c) To identify the scope of improvement in waste heat recovery system for smoke tube boiler.

4. METHODOLOGY

In the methodology, first the detailed study of smoke tube boiler is carried out. After that performance analysis of existing waste heat recovery boiler is done and analytical design of boiler is prepared. After analytical design, the CFD model in ANSYS FLUENT is developed and CFD results are validated.

5. ANALYTICAL DESIGN

Available data for calculations:-

1. Type of boiler – smoke tube boiler
2. Tube length (L) – 6m
3. Internal diameter of tube (Di) = 44.3mm
4. Outer diameter of tube (Do) = 50.8mm
5. Mass flow rate of flue gases (mh) = 0.53kg/sec
6. Mass flow rate of water=0.183kg/sec
7. Number of tubes in boiler (Nt) = 599
8. Inlet temperature of flue gas (Thi) = 310º C
9. Inlet (Tci) & Outlet temperature (Tco) of water = 30º C & 100º C
10. Specific heat of flue gas (Cph) =1.122kJ/kgK & specific heat of water(Cpc)= 4.183kJ/kgK

Heat available for waste heat recovery system

$$Q = (mh \times Cph \times \Delta T)\ \text{flue gases}$$
$$= 0.53 * 1.122 * (310-30)$$
$$= 166.50\ \text{kW}$$

Where,

Q =heat transfer rate (W)
mh= mass flow rate of flue gases (kg/sec.)
Cph = specific heat of flue gases (kJ/kgK)

By applying law of conservation of energy to boiler:-

Heat supplied by gas =Heat absorbed by water
(mh×Cph×ΔT) flue gases = (mc×Cpc×ΔT) water
0.53*1.122*(310-Tho) = 0.183*4.183*(100-30)
Tho = 220^0C

Amount of heat energy required for heating water
Q = 0.183*4.183*(100-30) = 53.58KW

Preliminary Analysis of Shell & Tube Exchanger(Boiler)
Following parameters value taken from TEMA Standard[8]

1. Single shell single pass Heat exchanger (CTP=.93)
2. Tube layout (CL=1)
3. Length of tube (L) =6m
4. Tube inside diameter (Di) =44.3 mm
5. Tube outside diameter (Do) =50.8 mm
6. Tube material steel (Kmt) = 26.1 W/mK
7. Pitch ratio (PR) =1.25
8. Baffle space=0.6Ds
9. Baffle cut=0.25Ds
10. hi=100 W/m²K ho=4000 W/m²K

A. Overall heat transfer coefficient
a. Without considering the fouling factor (Uo)

$$\mathrm{Uo} = \frac{1}{\frac{1}{\mathrm{hi}} * \frac{\mathrm{ro}}{\mathrm{ri}} + \ln\left(\frac{\mathrm{ro}}{\mathrm{ri}}\right)\left(\frac{\mathrm{ro}}{\mathrm{k}}\right) + \frac{1}{\mathrm{ho}}}$$

= 84.38W/m²K

b. With fouling factor
Rfo=0.0001754m²K/W

$$\mathrm{Uf} = \frac{1}{\frac{1}{\mathrm{hi}} * \frac{\mathrm{ro}}{\mathrm{ri}} + \ln\left(\frac{\mathrm{ro}}{\mathrm{ri}}\right)\left(\frac{\mathrm{ro}}{\mathrm{k}}\right) + \frac{\mathrm{ro}}{\mathrm{ri}} * \mathrm{Rf} + \frac{1}{\mathrm{ho}}}$$

= 69.82W/m²K

B. Properties of fluids

	Flue gas	water
Density (kg/m3)	0.6825	985
Specific heat (kJ/kgK)	1.109	4.183
Thermal conductivity (W/mK)	0.044	0.6513
Kinematic viscosity (m²/s)	$39.30 * 10^{-6}$	$0.478 * 10^{-6}$
Prandtl No.(Pr)	0.66	3.020

C. LMTD for counter flow heat exchanger (Boiler)

$$\Delta Tm = \frac{\Delta T1 - \Delta T2}{\ln\left(\frac{\Delta T1}{\Delta T2}\right)}$$

$= 200^0C$

1. Area of heat exchanger
   $Q = U*A*\Delta Tm$
   Area = 3.79m²

2. Shell side diameter (Ds)

$$Nt = 0.785 \frac{CTP}{CL} * \frac{Ds^2}{PR^2 * Do^2}$$

Ds = 1.81m

Where,
CTP = tube count calculations constant (1 tube pass = 0.93)
CL = tube layout constant (CL = 1)
PR = pitch ratio (1.25)

3. Baffles space=0.6*Ds=1.086m

4. Baffle cut=0.25*Ds=0.4525m

5. **Mc Adam equation,**

$$\frac{ho * De}{k} = 0.36 \left(\frac{De * Gs}{\mu}\right)^{0.55} * \left(\frac{Cp * \mu}{k}\right)^{0.33} * \left(\frac{\mu b}{\mu w}\right)^{0.14}$$

Where,
De- equivalent diameter of the shell side
K,µ and Cp- fluid properties on shell side at bulk mean temperature
Gs- shell side mass velocity

µw - viscosity at wall temperature
µb – viscosity at bulk mean temperature

$$De = \frac{4 * (Pt^2 - \frac{\pi}{4} * Do^2)}{\pi * Do}$$

= 0.050m

$$Gs = \frac{m}{As}$$

=0.53/0.03931
= 1.34kg/m²s

D. Shell side pressure drop

$$\Delta Ps = \frac{f * Gs^2 * (Nb + 1)Ds}{2 * \rho * De * \varphi s}$$

$$f = \exp\{0.576 - 0.19\ln(Re)\}$$

f = 0.4326

Where, f = friction factor
Nb = no. of baffles

ΔPs = 114.44N/m²

G. Tube side pressure drop

$$\Delta Pt = \frac{4 * Np * \rho * Uin^2}{2}\left(\frac{f * L}{Di} + 1\right)$$

$$At = \frac{\pi}{4} * Di^2 * \frac{Nt}{Np}$$

At = 0.9232m²

$$Uin = \frac{mt}{\rho * At}$$

Uin = 0.000198 m/s

ΔPt = 0.00460 N/m²

6. COMPUTATIONAL FLUID DYNAMICS (CFD) DESIGN OF HEAT EXCHANGER (BOILER)

Overview of CFD

Computational fluid dynamics (CFD) is a technique of predicting fluid flow, heat transfer, chemical reactions, mass transfer by solving the mathematical equations. The main application of CFD is as an engineering method, to provide data that is complementary to theoretical and experimental data. This is mainly the domain of commercially available codes and in-house codes at large companies. CFD can also be used for purely scientific studies, e.g. into the fundamentals of turbulence. This is more common in academic institutions and government research laboratories. Codes are usually developed to specifically study a certain problem.

CFD simulation of STHE

CFD simulation of STHE consists of following steps:-

1. Pre-processing
2. Solver set-up
3. Post processing
4. Validation

Geometry creation of model is done in SOLIDWORKS 2016. Geometry of model is created according to the specifications of STHE. After creation of geometry, meshing and discretization of geometry is done. The entire geometry is divided into two fluid domains shell side (flue gas) and tube side (water) in order to have better control over the nodes of elements. After meshing of model, model is imported to FLUENT where simulation is carried out. The model simulated has flue gas as one fluid and water as another fluid. The state of the various portions of the geometry such as fluid or solid is check in the cell zone conditions. Boundary conditions are selected for inlet and outlet of shell and tube.

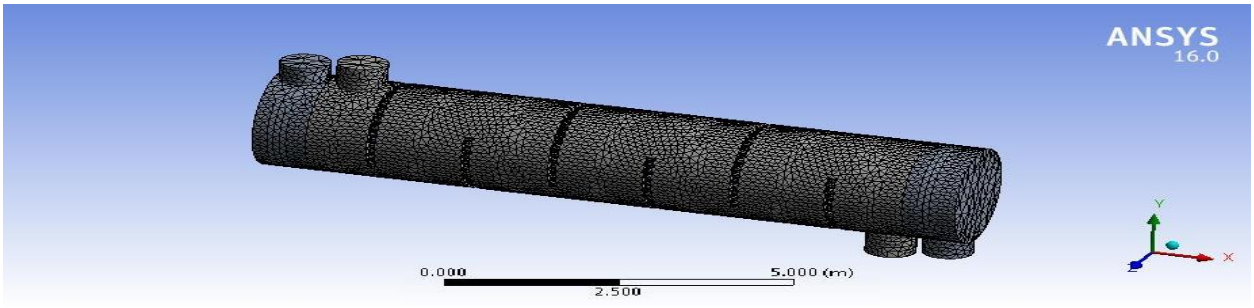

Fig.2.Meshing diagram of shell and tube heat exchanger

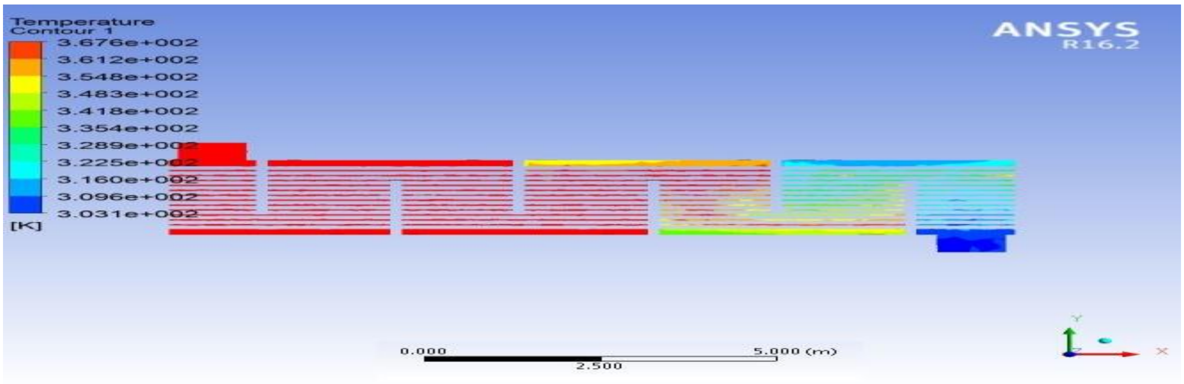

Fig.3.Contours of temperature for water domain(tube side)

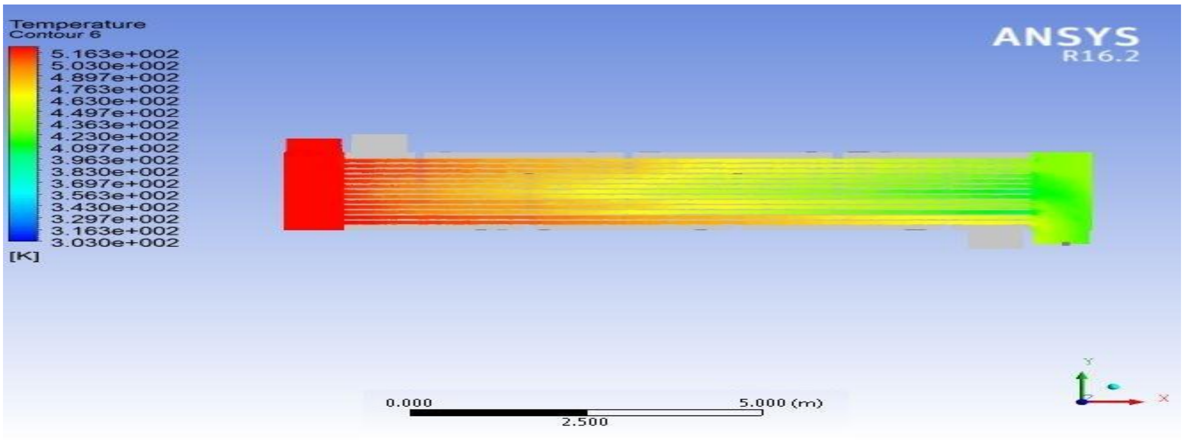

Fig.4.Contours of temperature for flue gas domain(shell side)

CFD simulation results

Fig.3. shows the contours of static temperature for water inside the tube side, in which blue color is the water inlet and orange color is the water outlet. From fig.3. it is seen that the temperature of water increases from 303K to 367K because of heat absorbed by water from the flue gases.

Fig.4. shows the contours of static temperature for flue gas inside the shell side, in which red color portion is the flue gas inlet and blue end is the flue gas outlet. From fig.4. it is seen that the temperature of flue gas drop down from 516K to 303K because of rejection of heat from flue gases to the water.

7. RESULTS AND DISCUSSION

Exit temperature of the flue gases from STHE in analytical analysis was obtained as 220℃, where in CFD simulation it was found to be 242℃. Thus, the percentage error of temperature for the flue gases between analytical calculation and CFD simulation is,

$$\%T_{\text{error}} = \frac{T_{\text{analytical}} - T_{\text{cfd}}}{T_{\text{analytical}}}$$

=10%

Exit temperature of the water from analytical analysis was obtained as 100℃, where in CFD simulation it was found to be 94.6

$$\%T_{\text{error}} = \frac{T_{\text{analytical}} - T_{\text{cfd}}}{T_{\text{analytical}}}$$

=5.4%

8. CONCLUSIONS

In the present work, performance analysis of smoke tube boiler for waste heat recovery is carried out. The CFD simulation of the waste heat recovery boiler is done. The temperature error of analytical and CFD for flue gas is 10% and for water 5.4%. So, to minimize the error fine meshing is required at very small surfaces, which are part of a larger surface, but where they rise to form an edge of a component, this will require very small meshing elements.

REFERENCES

[1] *Shubham Agarwal , Amit Suhane, 'Study of boiler maintenance for enhanced reliability of system a review', Journal of Materials Today: Proceedings Vol.4, pp.1542-1549.*

[2] *M.C. Barma, et al., 'A review on boiler energy use, energy savings, and emissions reductions', Journal of Renewable and Sustainable Energy Reviews, Vol.79,pp. 970-983.*

[3] *Ram Thakar, et al., 'Design of heat exchanger for waste heat recovery from exhaust gas of diesel engine', Procedia Manufacturing, Vol.20,pp.372-376.*

[4] *Elliot Woolley, et al., 'Industrial waste heat recovery: A systematic approach', Journal of Sustainable Energy Technologies and Assessments Vol.29,pp.50-59.*

[5] *M.A. Gomez, et al., 'Steady CFD combustion modelling for biomass boilers: An application to the study of the exhaust gas recirculation performance',Journal of Energy Conversion and Management, Vol.179,pp. 91-103.*

[6] *Ratchaphon Suntivarakorn, et al., 'Improvement of Boiler's Efficiency Using Heat Recovery and Automatic Combustion Control System', Energy Procedia Vol.100,pp.193-197.*

[7] *C P Kothandaraman, S Subramanyan, "Heat and mass transfer data book 8th edition" ,new age international publishers.*

[8] *Standards of Tubular Exchanger Manufacturers Association, 9th edition, 2007.*

[9] *Panagiotis Drosatos, et al., 'An in-house built code incorporated into CFD model for the simulation of boiler's convection section', Journal of Fuel Proceeding Technology, Vol.202,pp. 106333.*

[10] *A. Zargoushi, et al., 'CFD modeling of industrial cold box with plate-fin heat exchanger:Focusing on phase change phenomenon',International Journal of Heat and Mass Transfer.*

[11] *Chamil Abeykoon, 'Compact heat exchanger-Design and optimization with CFD', International Journal of Heat and Mass Transfer, Vol.146,pp. 118766.*

[12] *Wei-Wei Wang, et al., 'Experimental and numerical investigations of a radial heat pipe for waste heat recovery', Journal of Applied Thermal Engineering, Vol.154,pp. 602-613.*

[13] *M. Manickam, et al., 'CFD modelling of waste heat recovery boiler', Journal of Applied Mathematical Modelling, Vol.22,pp. 823-840.*

[14] *Rupesh Suryavanshi, et al., 'A Review on Waste Heat Recovery in Industries', International Journal of Research in Advent Technology, Vol.5, No.4, April 2017.*