Improvement in Thermal Performance of Boiler in Dairy Plant

Krunal A. Badwaik[1] Dr. P. V. Walke[2]
[1]M. Tech Student [2]Professor
[1,2]Department of Mechanical Engineering
[1,2]G.H. Raisoni College of Engineering, Nagpur, India

Abstract— Now a day's energy is very important. Energy saving is one of the important issues, not only from the point of view of fuel consumption but also for the protection of global environment. So, it is very important that a significant effort should be made for conserving energy through waste heat. In the present scenario most of the production process, industries consider boiler efficiency is a major factor. Due to industry demand engineer must continuously focus on improving the thermal performance of boiler. This project will address the problem of heat energy is wasted from dairy plant in the form of condensed water. The main object is to recover waste heat water from the system to recirculate to the economizer for efficiency improvement and enhance heat transfer rate. Analysis of existing economizer in dairy plant with modified condition in CFD (Computational Fluid Dynamics). To study the temperature and pressure inside the tube. In this paper simulation of economizer zone of boiler, from one of the dairy plant is consider economizer in preheat boiler feed water before enter into the boiler. Economizer use waste heat flue gases for steam production. Best ways of consumption of flue gases heat is transferring it to the boiler feed water. The model is solved using conventional CFD techniques by ANSYS software and simulation was carried out in ANSYS®FLUENT®v16. The Computational Fluid Dynamics (CFD) approached is utilised for the creation of three dimensional model of the economizer. With equilibrium assumption applied for description of the system chemistry. The condensed water temperature, pressure and velocity field of fluid flow within an economizer tube using the actual boundary conditions have been analyzed using CFD tool.
Keywords: Boiler; Economizer; CFD; Modelling

I. Introduction

Economizer is a mechanical device. Which helps in preheat boiler feed water before enter into the boiler. The word economizer comes from early use of such heat exchanger to reduce operating costs or economize on fuel by recovering extra energy from the flue gas. In simple word, economizer is a heat exchanger. A heat exchanger is a device that enables the transfer of heat from one fluid (gas or liquid) to another fluid between without direct contact.

In Boiler, economizer installed for use waste heat flue gases for steam production. Best way of utilization of flue gas heat is transmitting it to the boiler feed water. Hence economizer are designed in such a way that feed water is allowed to pass the economizer through pipe. Economizer shell is completely filled with the flue gasses. There is cross flow heat exchanger process takes place between boiler feed water and flue gas. Boiler feed water is heated up to its boiling temperature by consuming the heat from flue gas. Hence as the boiler feed water is already preheated this action of economizer helps in economic production of power plants. It reduces the operating costs of power plant.

Heat is considered to be a form of energy. So as the saying goes energy is neither be created not be destroyed. One of the ways increase the heat transfer rate of economizer. The heat energy is wasted from plant in the form of condensed water. We should recover waste heat in the form of condensed water from the system to recirculate to the economizer. Hence by mixing condensed water with boiler feed water to economizer. So we can increase the heat transfer rate and increase the temperature at outlet of economizer feed water. Analysis of existing economizer with modified condition in CFD (Computational Fluid Dynamics). The ultimate goal of economizer analysis is to achieve necessary heat transfer at minimum cost. CFD modelling is a good tool to improve the efficiency of economizer by analysis of mixing feed water with condensed water to economizer. CFD analysis of economizer can be useful to gain insight to the gas flow distribution. Efforts are made to measure the pressure and temperature distribution of flue gases at the economizer coil it will be useful to find the effect of the operating parameter on the tube pressure distribution inside the economizer. CFD has evolved as important tool for modelling of water tube boiler and it can useful to quantify the gas flow field and temperature distribution in the economizer. Hence ANSYS®FLUENT®v16 software was use to study the pressure and temperature distribution of the flue gases inside the economizer.

II. Computational Model for Economizer

A. *Problem Description:*

Analysis of exciting economizer with modified condition CFD. To study the temperature and pressure inside the tube.

B. *Computational Model:*

The computational model of an experimental tested economiser and the geometry parameters are listed in Table 1. Fig. 1. The whole computation domain is finite by the inner aspect of the shell and everything in the shell contained within domain. The inlet and outlet of the domain are connected with the corresponding tubes.

To simplify numerical simulation, some basic characteristics of the process following assumption are made:

1) The shell side fluid is constant thermodynamic properties.
2) The fluid flow and heat transfer processes are turbulent in nature.
3) The natural convection induced by the fluid density variation is neglected.
4) The tube wall temperature kept constant in the whole shell side.
5) The heat exchanger is adiabatic in nature.

Length of economiser	90 cm
Inner diameter of shell	45 cm
Inner diameter of tube	2.5 cm

Tube bundle geometry and pitch	2.5 cm
Number of tubes	26

Table 1: Geometric dimensions of shell and tube heat exchanger

III. GEOMETRY AND MESH

The model is designed according to existing economizer place in a plant.

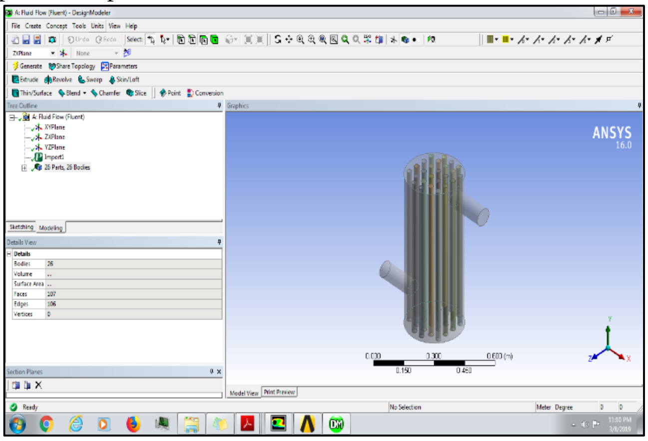

Fig. 1: Isometric view of Economizer

Sr. No.	Input Parameters	Value	Unit
1	Inlet temp of feed water	63	^{0} C
2	Outlet temp of feed water	119.3	^{0} C
3	Mass flow rate of feed water	0.035	Kg/sec
4	Inlet temp of flue gas	320	^{0} C
5	Outlet temp of flue gas	173.6	^{0} C
5	Mass flow rate of flue gas	1.1	Kg/sec

Table 2: Input parameter of CFD

IV. MESHING

Initially a relatively coarser mesh is generated with 2.0 Million cells. This grid contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use hexa as much as possible, for this reason the geometry is divided into several parts for using automatic methods available in the ANSYS meshing client. It is meant to decrease numerical diffusion to the degree that possible by structuring the mesh in a well manner, particularly near the wall region. Later on, for the mesh independent model, a fine mesh is generated with 5.8 Million cells. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

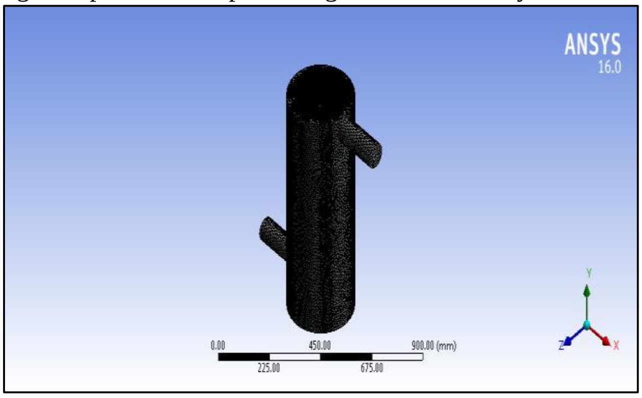

Fig. 2: Meshing diagram of shell and tube heat exchanger

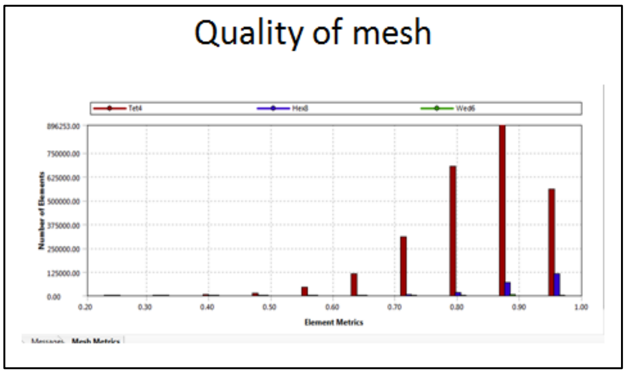

Fig. 3: Quality of mesh

V. PROBLEM SET UP

Simulation was carried out in ANSYS® FLUENT® v16. In the Fluent Solver Pressure Based type was particular, absolute velocity formation and steady time was selected for the imitation. In the model option energy calculation was on and the viscous was set as standard k-e, standard wall function (k-epsilon 2 equation). In cell zone fluid water-liquid and exhaust gas were selected. Water-liquid, exhaust gas and copper, aluminium was selected as materials for simulation. Boundary condition was selected for inlet and outlet.

For Flue Gas	
Mass flow rate (Kg/s)	1.1
Specific Heat (J/Kg K)	1120
Thermal Conductivity(W/mk)	0.460
Density (Kg/m^3)	1.337
Viscosity (Kg/m-s)	2.81e-5
For Feed water	
Mass flow rate (Kg/s)	0.035
Specific Heat (J/Kg K)	4200
Thermal Conductivity(W/mk)	0.141
Density (Kg/m^3)	913

Table 3. For Inlet Boundary Condition

VI. SOLUTION INITIALIZATION

Pressure velocity coupling selected as simple. Skewness correction was set at zero. In spatial discretization zone gradient was set as least square cell based, pressure was standard, momentum was second order upwind. Turbulent kinetic energy was set as second order upwind; energy was also set as second order upwind. In solution control, pressure was 0.8, Density 0.9, body force 1, momentum 0.1, turbulent kinetic and turbulent dissipation rate was set at 0.9, energy and turbulent viscosity was 0.8. Solution initialization was standard method and solution was initializing from inlet with 3298K temperature.

VII. RESULT

Under the above boundary condition and solution initialize condition simulation was set for 500 iteration.

A. Convergence of Simulation:

The convergence of simulation is necessary to get the parameters of the shell and tube heat exchanger in outlet. It also gives accurate value of parameters for the requirement of heat transfer rate. Continuity, X-velocity, Y-velocity, Z-velocity, energy, k-epsilon are the part of scaled residual which have to converge in a specific region. For the continuity, X-velocity, Y-velocity, Z-velocity, k-epsilon should be less than 10-4 and the energy should be less than 10^7. If these all values in same manner, then solution will be converged.

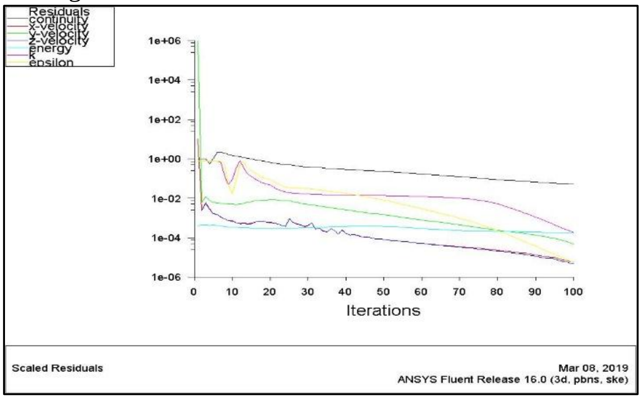

Fig. 4: Residual Plot

B. *Variation of Temperature:*

Temperature profile is examined to understand the flow distribution across the cross section at different positions in heat exchanger. The temperature at outlet of economizer feed water is 119.6 ^{O}C.

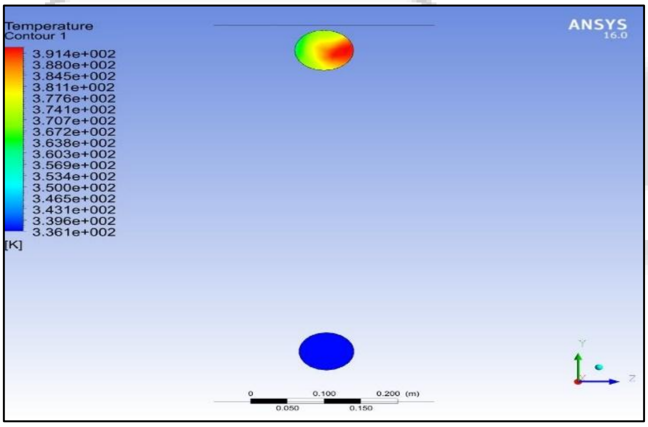

Fig. 5: Temperature of feed water at inlet and outlet of pipe

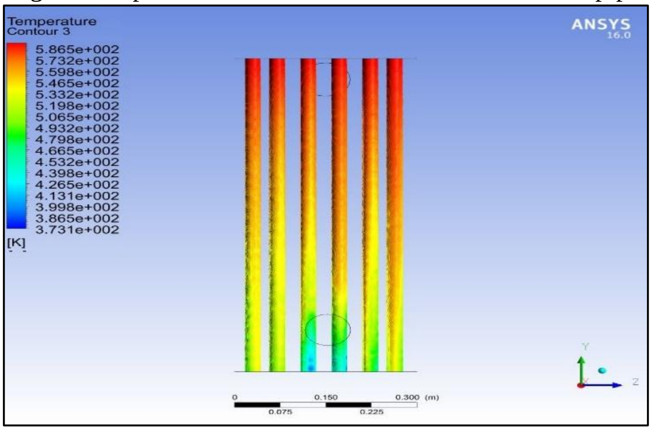

Fig. 6: Temperature distribution along the pipe

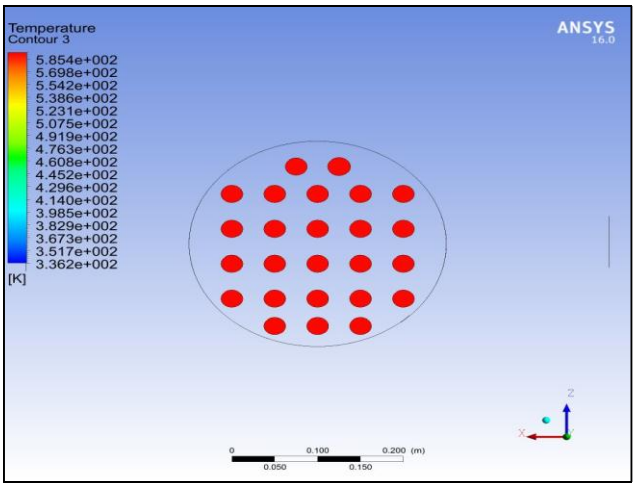

Fig. 7: Temperature at inlet of pipe (flue gas)

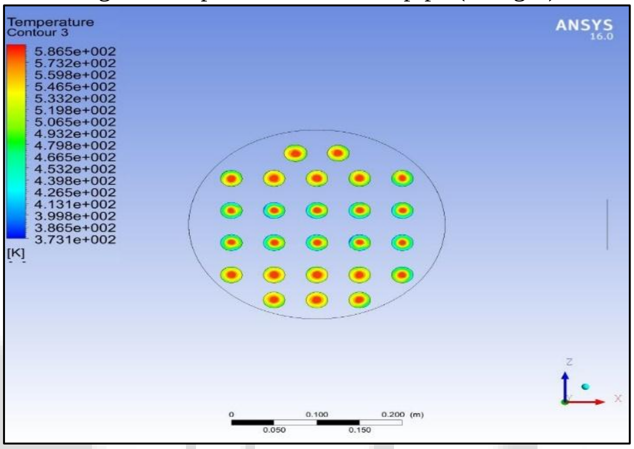

Fig. 8: Temperature at outlet of pipe (flue gas)

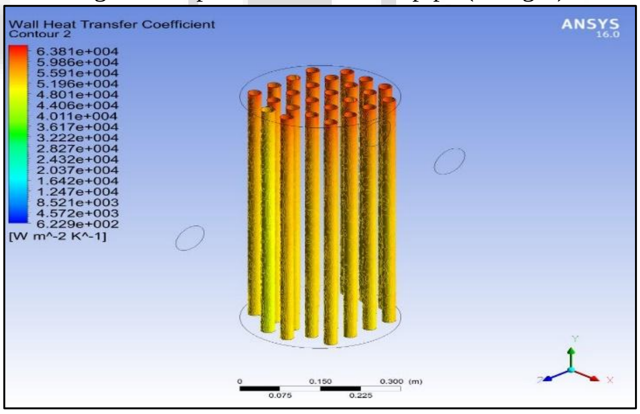

Fig. 9: Wall heat transfer coefficient along the tube

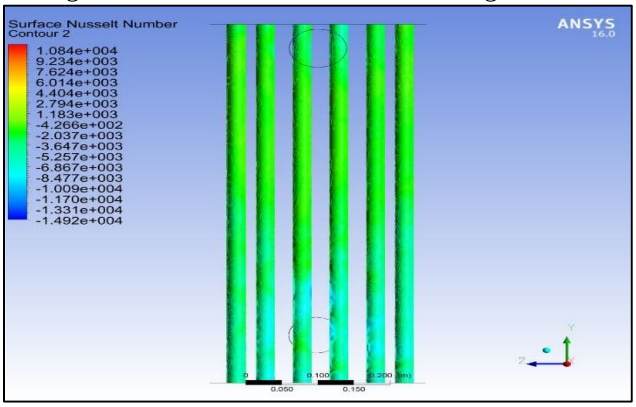

Fig. 10: Surface Nussle Number

VIII. CONCLUSION

In CFD analysis thermal performance by varying the location in the economizer and in turn improve the performance of boiler. The boiler feed water with mixing condensed water to economizer, increase temperature of feed water also its length as it picks up the heat from hot flue gas while moving towards outlet. The temperature profile reveals that the hot flue gas loses heat as it moves downward and the heat is gained by the feed water. The temperature of feed water at in the tube is less at inlet and high as it moves towards outlet.

REFERENCES

[1] Krunal P. Mudafale and Hemant S. Farkade, "CFD Analysis of Economizer In A Tangential Fired Boiler". Vol. 6, Issue 1, January 2017.
[2] K. Obual Reddy, M. Srikesh, M. Kranthi Kumar, V. Santhosh Kumar, 2014. "CFD Analysis of economizer to optimize heat transfer".International Journal Of Mechanical Engineering and Technology (IJMET). Volume 5, Issue 3, March (2014), pp. 66-76.
[3] Mukesh M. Chopade, 2018. "Thermal Design of Economizer for Waste Heat Recovery Boiler (WHRB)". Vol. 7 Issue 10, October-2018.
[4] R. PachaiyappAn, J. Dasa Prakash, 2015. "Improving the boiler efficiency by optimizing the combustion air". Applied Mechanics and Materials. August 2015 DOI: 10.4028/www.scientific.net/AMM.787.238.
[5] Nabil M. Muhaisen, Rajab Abdullah Hokoma, 2012. "Calculating the efficiency of steam boilers based on its most effecting factors: a case study".World Academy of Science, Engineering and Technology International Journal of Industrial and Manufacturing Engineering.
[6] K. Sampath Kumar Reddy, Dr B. Veerabhadra Reddy. "Performance of the Boiler and To Improving the Boiler Efficiency Using CFD Modeling". IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 8, Issue 6 (Sep. - Oct. 2013), PP 25-29 www.iosrjournals.org
[7] Nishadevi Jadeja, Sanjaysingh Zala, 2017, "Improved performance of a industrial packaged boiler by use of economizer". International Journal of Innovative ReseRch in Science Engineering and Technology. Vol. 6, Issue 1, January 2017.
[8] Madhav D., Ramesh L. & Naveen M., 2013. "Heat recovery through boiler blow down tank". International Journal Of Engineering Trends And Technology (IJETT).
[9] Tai, L., Zhenwei, G. & Yang, G. Power Plant Boiler EnergyWaste Heat Recycling Design Research.Power And Engineering Conference (Appeec), 2012 Asia-Pssacific, 2012. Ieee, 1-3.
[10] Mr. Amitkumar G. Tiwari, Mr. Shubham S. Jane, Mr. Pravin Katoke, Mr. Kunal Talmale, Mr. Kritesh Dongre, "Case Study on Efficiency of Boiler and Factors Affecting It". IJSRD - International Journal for Scientific Research & Development| Vol. 5, Issue 01, 2017 | ISSN (online): 2321-0613.