Numerical Modeling of an Economizer Using Serrated and Rough Surface for Heat Transfer Augmentation

Nagaraju H[1], K Lokesh[1], Hitesh BN[1], Dinesh K[1], Rajesh Babu.C[2]

School of Mechanical Engneering, REVA University, Bengaluru

Abstract—Economizers are used in conventional coal-fired thermal power plant to reduce the wastage of thermal energy through flue gases and increase the efficiency of boiler. This helps the industry as there is increase in profit. From the previous numerous research we have found that, the efficiency of economizer can be increased by using different passive, active and compound techniques. There are a number of passive techniques which can be used to enhance the heat transfer of an economiser. Some of the commonly used passive techniques are rough surfaces, treated surfaces, extended surfaces etc. Our project mainly concentrates on the serrated fins +Rough surface. There have been work on serrated fins extended surfaces in case of heat exchangers but less amount of work has been done in case of economizers. Our work concentrates on the twisting of blades of serrated fins and the effect it produces on the efficiency of economizer using CFD model. We will be comparing the CFD result with the industrial data.

Index Terms— Economizer, flue gases, heat exchangers, CFD.

◆

1 INTRODUCTION

Effective utilization of available energy becomes very important which is possible by using effective devising. When it is concerned with heat energy the devices are heat exchangers. A heat exchanger is thermal equipment, which is built for efficient heat transfer between two fluids of different temperature. The economizer is a mechanical device used to preheat boiler feed water before it enters the boiler drum by utilizing the waste heat captured from boiler flue gasses.It is absolutely necessary that this equipment be used to recover heat. Subsequently, this equipment has an important effect on energy use trends in each industrial sector

These techniques adopted typically increase the fluid mixing by increasing flow vorticity, turbulence or by limiting the growth of thermal boundary layer close to the heat transfer surface. Passive techniques are widely preferred to use for heat transfer enhancement due to the advantage of simplicity, easy to manufacture and cost-effectiveness.

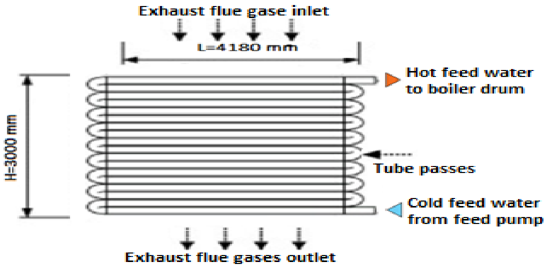

Fig.1. Schematic diagram of the economizer for multi-passcounter cross flow [2]

2 LITERATURE SURVEY

A. J. Mahmood ET, al [2019]: constructing and test single-pass and double-pass solar air heaters (SAHs) with four transverse fins. Demonstrate a significant improvement in the thermal efficiency and outlet air temperature.

Tian et al. [2015]: The comprehensive numerical analysis for fully developed forced convection heat transfer on a staggered circular pin fin. Studied on geometric parameters the fin diameter, fin height, and fin number have positive effects on heat transfer and negative effects on improving thermo hydraulic performance.

Bharadwaj et al. [18]determined pressure drop and heat transfer characteristics of the flow of water in a 75-start spirally grooved tube with twisted tape insert. It is found that the direction of twist (clockwise and anticlockwise) influences the thermo-hydraulic characteristics (Fig.2.5). Constant pumping power comparisons with smooth tube characteristics show that in spirally grooved tube with and without twisted tape, heat transfer increases considerably in laminar and moderately in the turbulent range of Reynolds numbers. However, for the bare spiral tube and for a spiral tube with anticlockwise twisted tape.

3 METHODOLOGY

The gas - water side heat transfer and pressure drop behaviour of an inline bare tube economizer investigated in a three-dimensional numerical study. An attempt was made to model and analyse the existing economizer using computational fluid dynamics (CFD) tool. The numerical results are compared with industrial data.

3.1 Problem Solving Approach in CFD

The basic steps involved in solving any CFD problem are as follows:

- Identification of flow domain.
- Geometry Modeling.
- Grid generation.

- Specification of boundary conditions and initial conditions.
- Selection of solver parameters and convergence criteria.
- Results and post processing.

The procedure to set-up and run a successful simulation in ANSYS Fluent, for a fluid flow problem, consists of a series of steps that are completed sequentially as outlined below

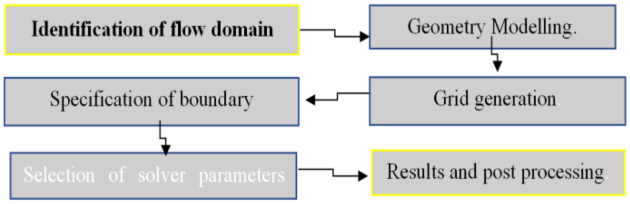

Fig:2Flowchart of CFD

a. Construction of the geometrical model using program such as SolidWorks.
b. Meshing of the fluid domain and setting up boundaries of the geometrical model into discrete volumes using appropriate meshing parameters and techniques via ANSYS Meshing. It is advantageous to have smaller volumes near the points of interest of the model and areas where the physical phenomena of the fluid will be more prevalent and important.
c. Determination and selection of the appropriate modelling technique available in ANSYS Fluent that best conforms to the conditions and phenomena of the flow situation of the problem.
d. Defining the boundary conditions and fluid properties.
e. Using the chosen solver in Fluent iterate for converged solutions of continuity, momentum, energy and turbulence.
Physical and computational domain:

This research investigates the influence of the existing design and operating parameters on fly ash particles deposition in heat exchanger tube bundles of water tube boiler - economizer. Ash deposition on heat transfer surface is a serious problem in economizer. In order to evaluate the influence of fly ash deposition on heat transfer tubes, a numerical analysis on heat transfer performance is carried out on a 96.8 t/h MCR (Maximum continuous rating) boiler unit. In this study, thermal performance is investigated using computational fluid dynamics (CFD) simulation using ANSYS Fluent 15. The fouling factor ε and the overall heat transfer coefficient ψ are employed to evaluate the influence of ash deposition. A numerical model has been developed based on the steady flow of flue gas, particle diameter and gas velocity which influence the deposition ratio. When ash particles impacts the tube wall, the deposition model have been implemented using descrite phase model (DPM) to determine the particle's state and discrete random walk (DRW) model is used to simulate the characterics of turbulent dispersion. The model demands significant computational details for geometric modeling, grid generation, and numerical calculations to evaluate the fly ash deposition on thermal performance of an economizer.

4 GOVERNING EQUATIONS

Governing fluid flow equations
To describe the fluid flow characteristics in the 3D computational domain, the following

Governing compressible fluid flow equations

Continuity equations:

$$\frac{\partial u_i}{\partial x_i} + \frac{\partial \rho}{\partial t} = 0 \ldots (1)$$

Momentum equation:

$$\rho\left[\frac{\partial u_j}{\partial t} + u_i \frac{\partial u_j}{\partial x_i}\right] = -\frac{\partial p}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_i} + \rho g_i \qquad (i,j = 1,2,3) \quad ..(2)$$

Energy equation

$$\rho\left[\frac{\partial h}{\partial t} + u_i \frac{\partial h}{\partial x_j}\right] = \frac{\partial}{\partial x_i}\left[\lambda \frac{\partial T}{\partial x_i}\right] - \tau_{ij}\frac{\partial u_j}{\partial x_i} + \frac{\partial p}{\partial t} + u_i \frac{\partial p}{\partial x_i} \ldots (3)$$

Here ϱ denotes the fluid density, t the time, u_i the velocity component in i direction and x_i the Cartesian coordinate, p is the pressure, g_j is the gravitational acceleration, h is the specific enthalpy, T is the temperature and λ is the thermal conductivity coefficient. Tij stands for the viscous stress tensor.

4.4.2 Governing equations for the incompressible fluid flow equations

To describe the steady state fluid flow characteristics in the 3D computational domain.

Continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \ \ldots\ldots\ldots\ldots\ldots\ldots\ldots (4)$$

From the first law of thermodynamics, the energy equation is derived as the net rate of increase in energy, this equals the net rate of heat added to fluid plus the net rate of work on the fluid. When the fluid flow is considered, the thermal energy is much higher than the kinetic or potential energy.

Energy 4.4.3 for an incompressible flow under steady state conditions, the equations

$$\frac{\partial(u_i[\rho e+p])}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\lambda_{eff}\frac{\partial T}{\partial x_i}\right) \ldots\ldots\ldots\ldots\ldots\ldots(5)$$

The heat conduction equation solved in the solid is presented as:

$$\frac{\partial(\rho s h)}{\partial t} - \frac{\partial}{\partial xi}\left(\lambda s\frac{\partial T}{\partial xi}\right) = 0 \ldots\ldots\ldots\ldots\ldots(6)$$

Where $h=C_sT$. Here, ϱ_s, C_s and $\lambda_s$are density, specific heat and the possibly anisotropic conductivity of the solid.

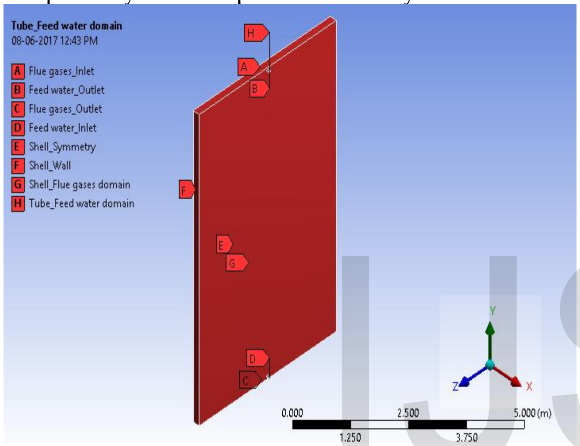

Fig.3 Simulation model with the boundary conditions adopted for the economizer

Grid dependency is the term used to describe the improvement of results by using successively smaller cell sizes for the calculations. A calculation should approach the correct answer as the mesh becomes finer, hence the term grid convergence. The normal CFD technique is to start with a coarse mesh and gradually refine it until the changes observed in the results are smaller than a pre-defined acceptable error. However, the developed algorithm that diverges very little as the cell size is increased making it much easier to obtain the necessary **CONVERGENCE_1983803** cells

1) Residual RMS Error values have reduced to an acceptable value (typically 10^{-3})
2) Monitor points for our values of interest have reached a steady solution.

Fig 4 Grid used for the simulation model

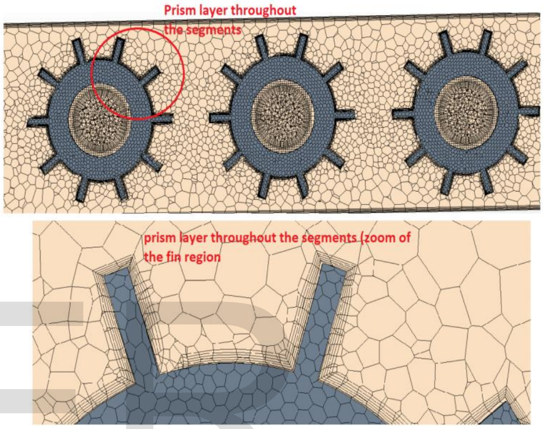

Fig 5 Grid used for the simulation model with zoom of the fin region

Fig:6 RMS residual values

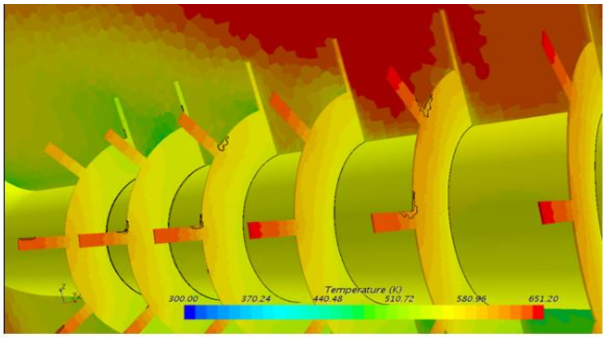

Fig:7 Temperature counters

5 RESULTS AND DISCUSSIONS

The simulation is defined for the shell and tube economizer with the mass flow, static Temperature at the inlet ans average static pressure at the exit. The steady state solution is obtained up to a convergence of le-5 with high-resolution. The temperature streamline plots are obtained for complete one tube row and 20 numbers of tubes in a row.

It can be obtained that the temperature difference and pressure drop of simulation result between the inlet and outlet flue gas are not in agreement with the industrial values. This is mainly due to the fact that size of the mesh quality.

The moniter convergence with residual alues 10^-5 has reached tolerance. But after reaching specified residual value the solver starts oscillating. Further for better accuracy run the CFD simulation using second order discretization. However the results presented in this report still under investigation. Some changes may still place to attend required results by using grid independent. These results consider as a case study under investigation.

6 FUTURE WORK

Future scope of work as been carried out by aasuming the fin parameters
1. Twisting of the blades of serrated fins.
2. Meshing of the economizer for 1 tube row.
3. Analysis of the meshed row and comparison with the data

7 REFERENCE

1. Johnson, I., Choate, W.T., and Davidson, A., Waste Heat Recovery. Technology and Opportunities in US Industry. 2008, BCS, Inc., Laurel, MD (United States).
2. Ross Brindle, J.E., Amanda Greene, Improving the Thermal Efficiency of Coal-Fired Power Plants in the United States. 2010: Sheraton Baltimore City Center Baltimore, MD.
3. Beck, M.P., Thermal conductivity of metal oxide nanofluids. 2008: Georgia Institute of Technology.
4. Ganapathy, V., Industrial boilers and heat recovery steam generators: design, applications, and calculations. 2002: CRC Press.
5. Beychok, M. Steam generator. 2013
6. Pongsoi, P., Pikulkajorn, S., and Wongwises, S., Heat transfer and flow characteristics of spiral fin-and-tube heat exchangers: A review. International Journal of Heat and Mass Transfer, 2014. 79: p. 417-431.
7. Mu, L., Zhao, L., and Yin, H., Modelling and measurements of the characteristics of ash deposition and distribution in a HRSG of wastewater incineration plant. Applied thermal engineering, 2012. 44: p. 57-68.
8. Bergles, A.E., High-flux processes through enhanced heat transfer. 2003.
9. Li, J., Du, W., and Cheng, L., Numerical simulation and experiment of gas-solid two phase flow and ash deposition on a novel heat transfer surface. Applied Thermal Engineering, 2017. 113: p. 1033-1046.
10. Bozzoli, F., Cattani, L., and Rainieri, S., Effect of wall corrugation on local convective heat transfer in coiled tubes. International Journal of Heat and Mass Transfer, 2016. 101: p. 76-90.
11. Stein, J., Waterside Economizing in Data Centers: Design and Control Considerations. ASHRAE Transactions, 2009. 115(2).
12. Liu, S. and Sakr, M., A comprehensive review on passive heat transfer enhancements in pipe exchangers. Renewable and sustainable energy reviews, 2013. 19: p. 64-81.
13. Bergles, A.E., Techniques to enhance heat transfer. 1998, New York: McGraw-Hill.
14. Promvonge, P., Pethkool, S., Pimsarn, M., and Thianpong, C., Heat transfer augmentation in a helical-ribbed tube with double twisted tape inserts. International Communications in Heat and Mass Transfer, 2012. 39(7): p. 953-959.
15. Thianpong, C., Eiamsa-Ard, P., Wongcharee, K., and Eiamsa-Ard, S., Compound heat transfer enhancement of a dimpled tube with a twisted tape swirl generator. International Communications in Heat and Mass Transfer, 2009. 36(7): p. 698-704.
16. Promvonge, P. and Eiamsa-ard, S., Heat transfer behaviors in a tube with combined conical-ring and twisted-tape insert. International Communications in Heat and Mass Transfer, 2007. 34(7): p. 849-859.
17. Pal, S. and Saha, S.K., Experimental investigation of laminar flow of viscous oil through a circular tube having integral axial corrugation roughness and fitted with twisted tapes with oblique teeth. Heat and Mass Transfer, 2015. 51(8): p. 1189-1201.
18. Bharadwaj, P., Khondge, A., and Date, A., Heat trans-

fer and pressure drop in a spirally grooved tube with twisted tape insert. International Journal of Heat and Mass Transfer, 2009. 52(7): p. 1938-1944.
19. Mazumder, A.K. and Saha, S.K., Enhancement of thermohydraulic performance of turbulent flow in rectangular and square ribbed ducts with twisted-tape inserts. Journal of Heat Transfer, 2008. 130(8): p. 081702.
20. Gong, B., Wang, L.-B., and Lin, Z.-M., Heat transfer characteristics of a circular tube bank fin heat exchanger with fins punched curve rectangular vortex generators in the wake regions of the tubes. Applied Thermal Engineering, 2015. 75: p. 224-238.