International Journal of Mechanical and Industrial Engineering

Volume 3 | Issue 2 Article 14

October 2013

CFD ANALYSIS OF ECONOMIZER IN A TENGENTIAL FIRED BOILER

KRUNAL P. MUDAFALE
Govt. College of Engineering, Amravati, Maharashtra, India, krunalp.mudafale@gmail.com

HEMANT S. FARKADE
Govt. College of Engineering, Amravati, Maharashtra, India, hemantsfarkade@gmail.com

CFD ANALYSIS OF ECONOMIZER IN A TENGENTIAL FIRED BOILER

KRUNAL P. MUDAFALE[1] & HEMANT S. FARKADE[2]

[1,2]Department of Mechanical Engineering, Govt. College of Engineering, Amravati, Maharashtra, India
E-mail : krunalp.mudafale@gmail.com, hemantsfarkade@gmail.com

Abstract - This paper presents a simulation of the economizer zone, which allows for the condition of the shell-side flow and tube-side and tube-wall, thermal fields, and of the shell-tube heat-exchange. Selection of the economizer zone from the thermal power plant only because, it is found trends of failure that the economizer is the zone where the leakages are found more. The maximum number of cause of failure in economizer unit is due to flue gas erosion. The past failure details revels that erosion is more in U-bend areas of Economizer Unit because of increase in flue gas velocity near these bends. But it is observed that the velocity of flue gases surprisingly increases near the lower bends as compared to upper ones. The model is solved using conventional CFD techniques by STAR- CCM+ software. In which the individual tubes are treated as sub-grid features. A geometrical model is used to describe the multiplicity of heat-exchanging structures and the interconnections among them. The Computational Fluid Dynamics (CFD) approach is utilised for the creation of a three-dimensional model of the economizer coil. With equilibrium assumption applied for description of the system chemistry. The flue gas temperature, pressure and velocity field of fluid flow within an economizer tube using the actual boundary conditions have been analyzed using CFD tool. Such as the ability to quickly analyse a variety of design options without modifying the object and the availability of significantly more data to interpret the results. This study is a classic example of numerical investigation into the problem of turbulent reacting flows in large scale furnaces employed in thermal power plants for the remediation of ash deposition problems. And the experimental setup is from Chandrapur Super Thermal Power Station, Chandrapur having the unit no IV of 210 MW energy generations.

Keywords*-Tangential fired boiler; Economizer; CFD; Modelling; corrosion;*

I. INTRODUCTION

Economizer performs a key function in providing high overall boiler thermal efficiency by recovering the low level energy from the flue gas before it is exhausted to the atmosphere. Economizer recovers the energy by heating the boiler feed water. Economizer are basically tubular heat transfer surfaces used to preheat boiler feed water before it enters the steam drum or furnace surfaces. Economizer reduces operating costs or economies on fuel by recovering extra energy from the flue gas. The ultimate goal of economizer design is to achieve necessary heat transfer at minimum cost. A key design criterion for economizer is maximum allowable flue gas velocity. Higher velocity provides better heat transfer and reduces capital cost. CFD modeling is a good tool to improve the efficiency of economizer by reducing the number of tubes of existing model. Duct having rectangular cross sectional area is normally used in power plant. Gas flow distribution over heating equipment is very critical problem in power plant. By providing inlet guide vanes at economizer inlet duct, we get effective heat transfer in economizer.[1] The strategy of how to recover this heat depends in part on the temperature of the waste heat gases and the economics involved. Large quantity of hot flue gases is generated from Boilers, Kilns, Ovens and Furnaces. If some of this waste heat could be recovered, a considerable amount of primary fuel could be saved. The energy lost in waste gases cannot be fully recovered. It was found from the trends of failure for most of the thermal power plants in India that the economizer is the zone where the leakages are found more. The maximum number of cause of failure in economizer unit is due to erosion. Erosion is a process in which material is removed from the surface layers of an object impacted by a stream of abrasive particles. CFD analysis of economizer can be useful to gain insight to the gas flow distribution. Efforts are made to measure the velocity and Temperature distribution of flue gases at the bends of the economizer coil it will be useful to find the effect of the operating parameter on the tube erosion rate and velocity and pressure distribution inside the economizer. CFD has evolved as important tool for modeling of coal fired boiler and it can useful to quantify the gas flow field and temperature distribution with the boiler economizer. Hence STAR- CCM+ software was use to study the velocity and temperature distribution of the flue gases inside the economizer.

II. EXPERIMENTAL SET UP

A. Specification of Economizer Coil

- Material: SA 210 Gr (carbon steel)
- Length of coil :8200 mm
- Pith of coil :115 mm
- Diameter: 44.5 mm
- Thickness : 4.5 mm
- No of turns : 9
- Turn diameter : 28.75 mm

B. Geometry Of Economizer Coil

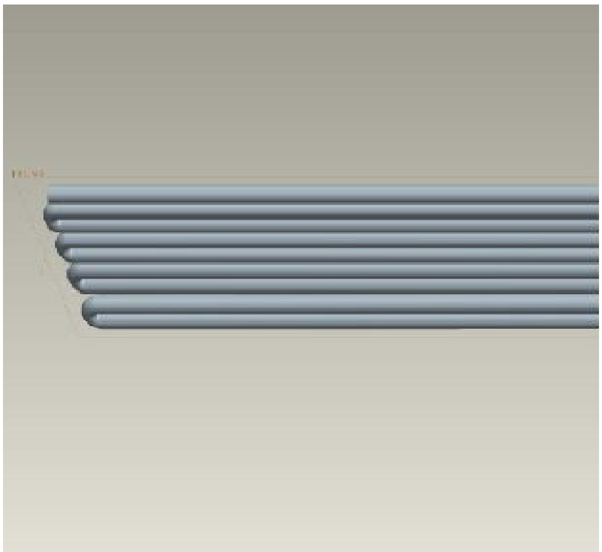

Figure 1: Pro-e model of Economizer coil
Geometry of the economizer coil made with the help of Pro-e 4.0.in figure No.1 then it is then saves in Iges file and import in star-ccm+ software directly.

III. CFDMODELLING OF ECONOMIZER ZONE

A Assumptions For The CFD Analysis Of Economizer Tube

- All the physical properties of the fluid and of the Solid are considered as constant.
- Steady and turbulent flow of fluid.
- The flow enters at constant temperature and with Uniform velocity.
- Model of turbulence adopted is k-ε. The model is drawn as per the actual geometry in figure2.

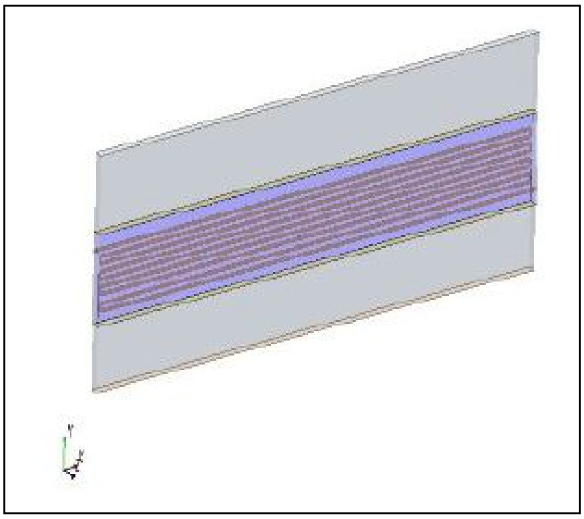

Figure 2: CFD Model of Economizer Coil

Sr. No.	Input Parameters	Value	Unit
1	Inlet temp of feed water	247	° C
2	Outlet temp of feed water	283	° C
3	Mass flow rate of feed water	1.359	Kg/sec
4	Inlet temp of flue gas	477	° C
5	Outlet temp of flue gas	366	° C
6	Mass flow rate of flue gas	1.9074	Kg/sec

These readings are for maximum control rating for generating 210 MW units.

IV. MESHING

A. Polyhedral Meshing

Polyhedral meshes provide a balanced solution for complex mesh generation problems. They are relatively easy and efficient to build, requiring no more surface preparation than the equivalent tetrahedral mesh. They also contain approximately five times fewer cells than a tetrahedral mesh for a given starting surface. Multi-region meshes with a conformal mesh interface are allowed. The polyhedral meshing model utilizes the arbitrary polyhedral cell shape in order to build the core mesh. In STAR-CCM+, a special dualization scheme is used to create the polyhedral mesh based on an underlying tetrahedral mesh which is automatically created as part of the process. The polyhedral cells created typically have an average of 14 cell faces. The polyhedral core mesh density can be increased or decreased by using the volume mesh density factors. Volumetric controls can also be included to locally increase or decrease the mesh density based on a range of prescribed shapes. When the polyhedral mesher is used in conjunction with a periodic interface, then it will attempt to produce a conformal mesh for the boundary pair where possible.

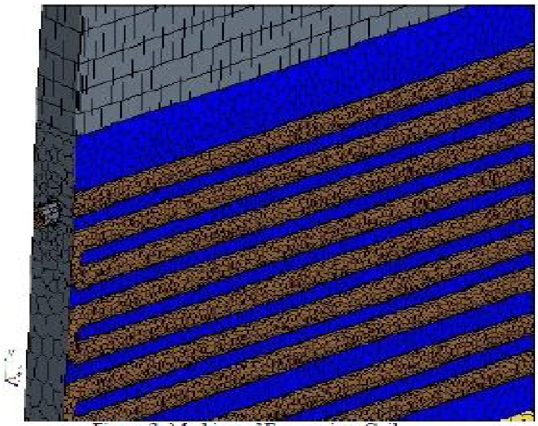

Figure 3: Meshing of Econcmizer Coil

V. CONVERGENCE

Residual monitor plots are very useful for judging the convergence (or divergence) of a solution, and they are created automatically within every simulation. However, it is important to understand both the significance of residuals and their limitations. While it is true that the residual quantity will tend toward a very small number when the solution is converged, the residual monitors cannot be relied on as the only measure of convergence. The limitations of residuals are as follows:

- The amount a residual decreases depends on the particulars of the simulation. Therefore, a three-orderof- magnitude drop in residuals might be acceptable for one simulation but not another. The initial guess also strongly influences the amount that residuals are reduced--if the initial solution satisfies the discretized equations perfectly, the residuals will not drop at all.
- Residuals do not necessarily relate to quantities of engineering interest in the simulation such as integrated forces, pressure losses or mass flow rates

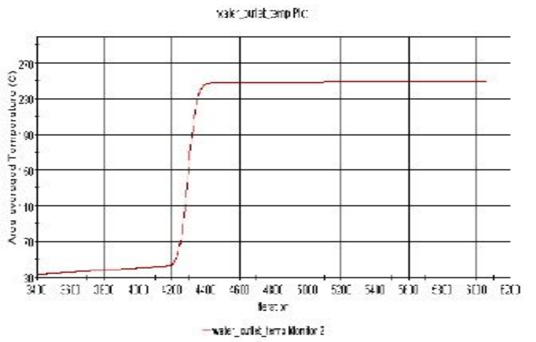

Figure 4: Convergence Plot

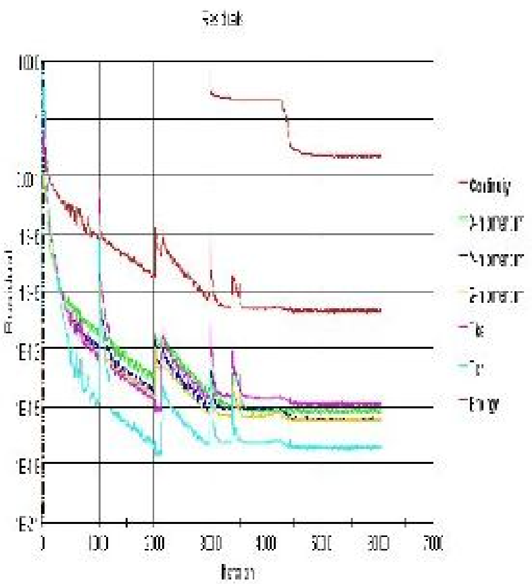

Figure 5: Residuals Plot

VI. RESULT ANALYSIS

A. Temperature Distribution in Economizer

Figure3, Figure4. Shows the temperature distribution of flue gas, feed water in economizer in a vertical plane along its length in X-Y plane. The temperature plot reveals that the hot flue gas looses heat as it moves downward and the heat is gained by the feed water. Inside the tube, the temperature of feed water increases along its length as it picks up the heat from hot flue gas while moving towards outlet.

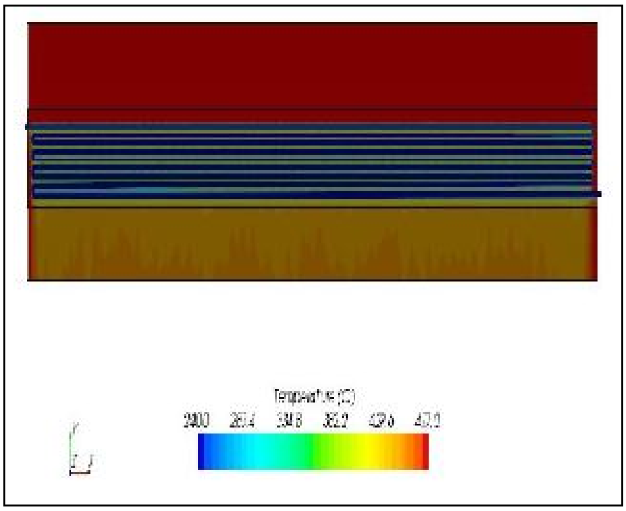

Figure 6: for flue gas

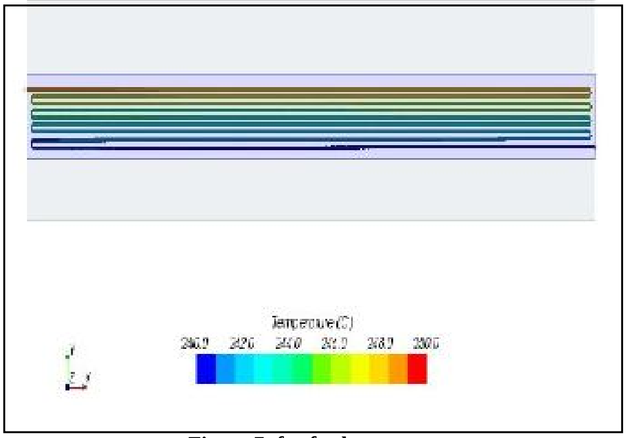

Figure 7: for feed water

B Velocity Distribution in Economizer

Figure.5, Figure 6 shows the Velocity distribution of flue gas in economizer in a vertical plane along its length in X-Y plane. The velocity plot reveals that the hot flue gas looses heat as it moves downward and the heat is gained by the feed water. Inside the tube, the velocity of feed water increases along its length as it picks up the heat from hot flue gas while moving towards outlet. Most of the air passes through the tubes, and a part of fluid moves downward.

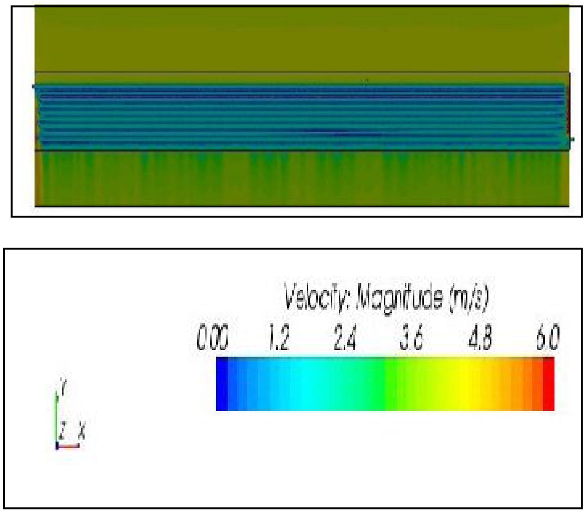

Figure 8: for flue gas

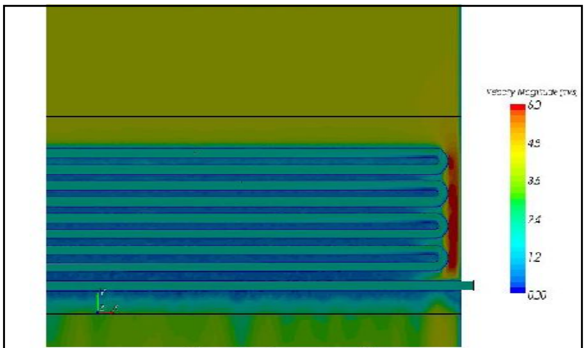

Figure 9: enlarge view of flue gas

C Pressure distribution in Economizer

The pressure contour of economizer in a vertical plane along its length in X-Z plane is shown in Figure7. The high pressure zone has been observed on the upper side of economizer coil, which may be due to large change in the momentum of feed water. The pressure of feed water at in the tube is higher at inlet and decreases as it moves towards outlet. The high pressure gradient along the tube length is due to friction between feed water and the tube wall.

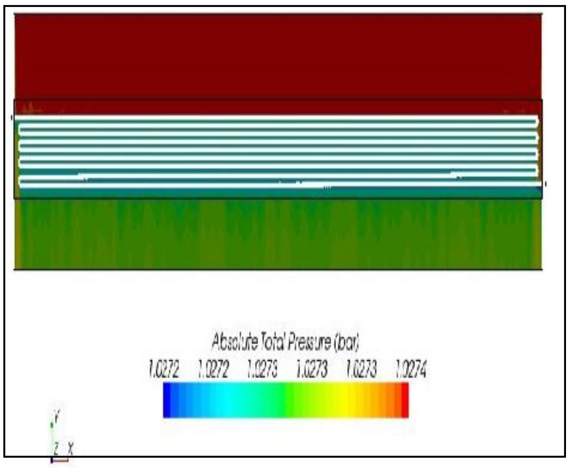

Figure 10: for flue gas

CONCLUSION

A method for predicting the pressure, velocity & temperature distribution in the tube type economizer associated with tangentially fired boiler developed by using SATR-CCM+ software. The CFD model may be used to optimize its thermal performance by varying the location in the economizer and in turn improve the performance of boiler. The temperature and velocity fields are in agreement with the expected behavior of a tangentially fired coal combustion chamber. The temperature of feed water increases along its length as it picks up the heat from hot flue gas while moving towards outlet. The velocity plot reveals that the hot flue gas looses heat as it moves downward and the heat is gained by the feed water. The pressure of feed water at in the tube is higher at inlet and decreases as it moves towards outlet. The high pressure gradient along the tube length is due to friction between feed water and the tube wall. In this study of the economizer the critical zone has been seen at the turning bend of the coil.

ACKNOWLEDGEMENT

The author is grateful to tkank the staff of Governent College of Engineering, Amravati.for their support and the experimental set up is of Chandrapur Super Thermal Power Station of unit no. IV of 210 Mw energy generation.

REFERENCE

[1] D. Patil,P. R. Baviskar, “Optimization of Economiser Design for the Enhancement of Heat Transfer Coefficient”

[2] J B Kitto and S C Stultz “Steam – its generation and use” The Babcock &Wilcox Company, 40th edition

[3] Charles E Baukal “Combustion Handbook”, the John Zink Company. [4] John D Anderson “Computational Fluid Dynamics the basics and application” McGraw-Hill,Dept. of Aerospace Engineering, University of Maryland.

[4] Suhas V Patankar “Numerical Heat Transfer and Fluid Dynamics”.

[5] P.K.NAG“Power Plant Engineering”, 2006, Tata McGraw-Hill Publication.

[6] PMV Subbarao “CFD Analysis of Flue Gas Duct”, 05 Jan 2010, IIT, Delhi.

[7] V Ganapathy “Industrial Boilers and Heat Recovery Steam Generators, CRC”, ABCO industries, Abilene, Texas, USA.

[8] S. Jayanti “Techniques for flow passage optimization in air and flue gas ducting in boilers”,BHEL National conference on CFD Application ,Hyderabad, 17-18 Nov,2006.

[9] Yogesh Jaluria “Design and Optimization of Thermal System”, Rutgers,The state University of New Jersey.

[10] STAR-CCM+ software User Guide.

[11] Hryb,D,, Cardozo, M., Ferro, S., and Goldschmit M., 2009 "Particle transport in turbulent flow using both lagrangian and Eulerian formulations",Int. Commun.Heat Mass Transfer.

[12] N. hedge, I. Han, T.W. Lee, R.P.Roy. (2007),"Flow and heat transfer in Heat Recovery Steam Generator". ASME Vol.129

[13] P.J. Stopford," Recent applications of CFD modeling in the power generation and combustion industrie", Appl.math. Modell 26(2002)

[14] Al-Khalidy Neihad,"Design optimization Of industrial ducts using CFD", CSIRO, 10-12 Dec, 2003, Australia.

[15] M.EI Sayed et al (2005)-"Shape optimization with computational fluid dynamics",Advances in Engg Software.

◈◈◈