ANALYSIS OF ECONOMISER USING THE CFD TOOL

Prof. Abhay Bendekar[1], Prof. Magesh Kumar[2], Prof. Shivram Poojari[3]

[1,2,3]*Asst. Professor, Mechanical Engineering , Shree L. R.Tiwari College of Engg. ,*
Thane (E), (India)

ABSTRACT

This paper presents a simulation of the economizer zone, which allows studying the flow patterns developed in the fluid, while it flows along the length of the economizer. 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 FLUENT 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 inter connections among them. The Computational Fluid Dynamics (CFD) approach is utilized for the creation of a three-dimensional model of the economizer coil of single column tube. 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 analysed using CFD tool.

This study is a classic example of numerical investigation into the problem of turbulent flows in U bends for the pressure drop and velocity variation in the flow so that it helps in design the economizer with low pressure losses for the thermal power plants.

Keywords: CFD, Economizer, FLUENT, flow efficiencies, , fluid dynamics, pressure drops.

I. INTRODUCTION

In boilers, economizers are heat exchange devices that heat fluids, usually water, up to but not normally beyond the boiling point of that fluid. Economizers are so named because they can make use of the enthalpy in fluid streams that are hot, but not hot enough to be used in a boiler, thereby recovering more useful enthalpy and improving the boiler's efficiency. Using an economizer can increase feed water temperature and reduce the amount of heat required in a boiler. The amount of heat that can be transferred and the upper limit of feed water temperature depend primarily on boiler (steam) pressure and temperature of flue gases discharged from the boiler. Transferring heat from the flue gases to the feed water will lower flue gas temperature. 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. The control over the fluid flow is absolute to increase the efficiency of the economizer and CFD modelling is a good tool to study the fluid flow, to improve the efficiency of economizer by reducing the number of tubes of existing model. 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. CFD has evolved as important tool for modelling of coal fired boiler and it can useful to quantify the fluid flow field and pressure distribution with the boiler economizer. Hence FLUENT software was used to study the velocity and pressure distribution of the working fluid inside the economizer.

II. DESCRIBING THE MODEL AND SIMULATIONS

A three dimensional model of an economizer is model with the standard specifications and dimensions from an industry, by using the software tool ICEM CFD. In the ICEM CFD, the economizer was modelled and meshed with the tetrahedral scheme. For this analysis, single unit of economizer is considered and modelled to observe the flow phenomenon and pressure drop in each step tube and the overall pressure drop of the economizer unit. It consists of 12 parallel pipes of 30 meters long and 23 C-shaped connecting tubes on either side of the parallel tube as shown in fig1.

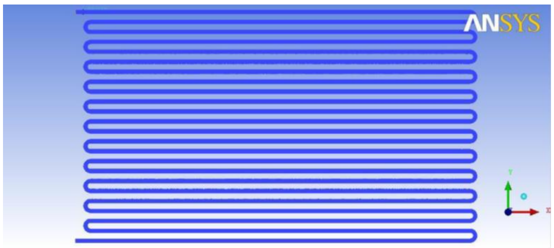

Fig. 1

III. MESHING & SOLVING

The modelled geometry was under gone discretization process, with the help of Mesh tool available in the ICEM CFD tool. On meshing the geometry in the ICEM CFD tool, it was observed to had quality of 0.72 in the relevance standards of the ICEM CFD tool. With the satisfactory mesh quality, it is found that it has:

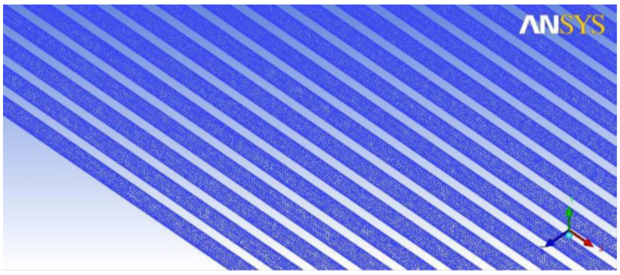

Fig. 2

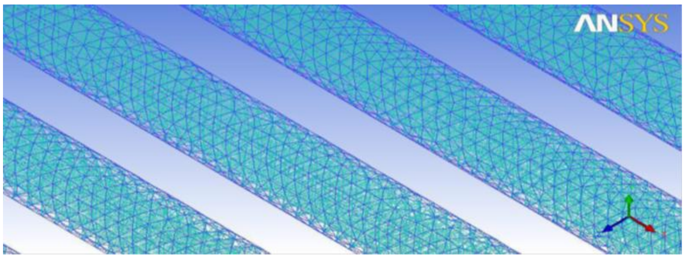

Fig. 3

ISBN: 978-81-932074-4-4

IV. BOUNDARY CONDITIONS

On discretising the geometry, the specific boundary conditions should be assigned to the surface of the elements, which decides the behaviour of the element to the solver. The following working and boundary conditions are to evaluate the performance between the right angled and circular tube pipes. In the fluent solver, the following boundary and solver conditions are used:

Model: Solve under energy equation with viscosity k-epsilon equation with standard wall function, with the possibility of viscous heating.

Material: Steel for the pipe wall and water vapour for the volumetric domain (fluid).

Boundary conditions: Inlet as mass flow rate as 71.5572 kg/s; with inlet pressure as 450000; with temperature 531 K; and with turbulent kinetic energy 1 m^2/s^2; turbulent dissipation rate 1 m2/s3; outlet as default outflow conditions and the wall temperature as 300 K.

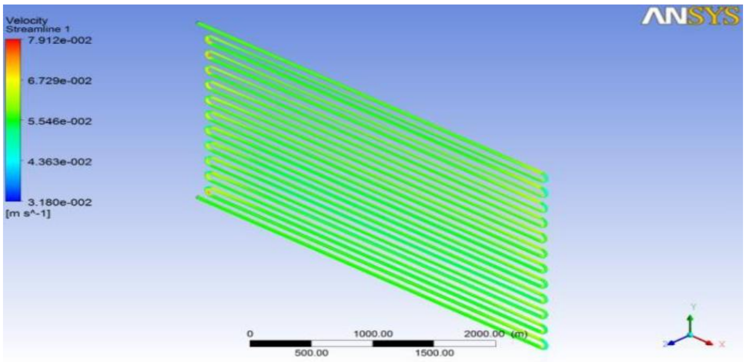

Fig. 4

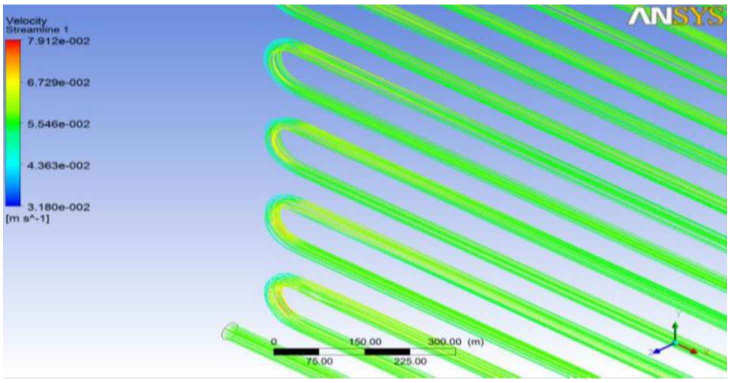

Fig. 5

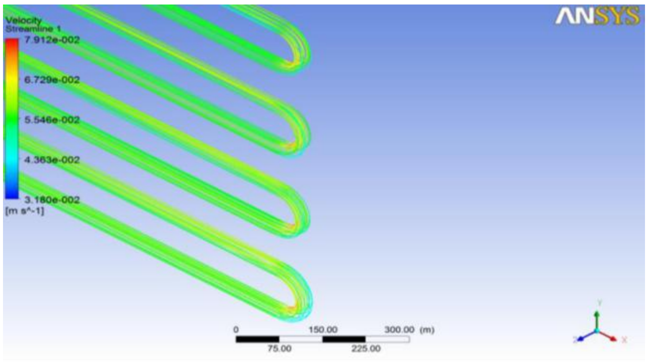

Fig. 6

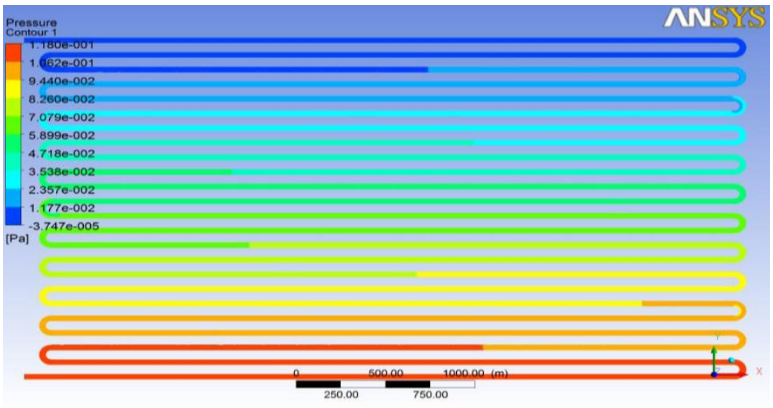

Fig. 7

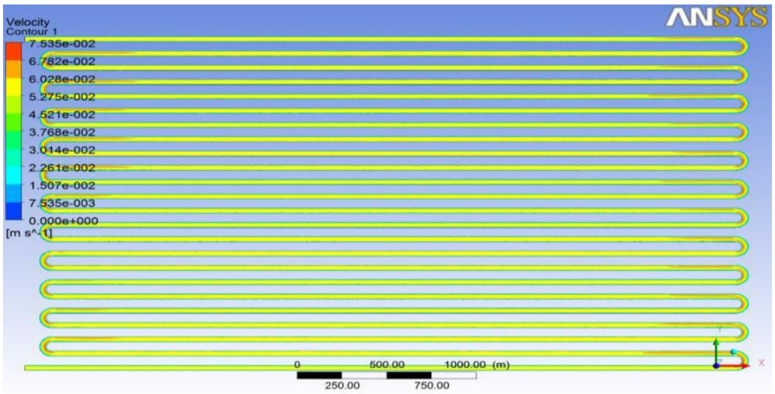

Fig. 8

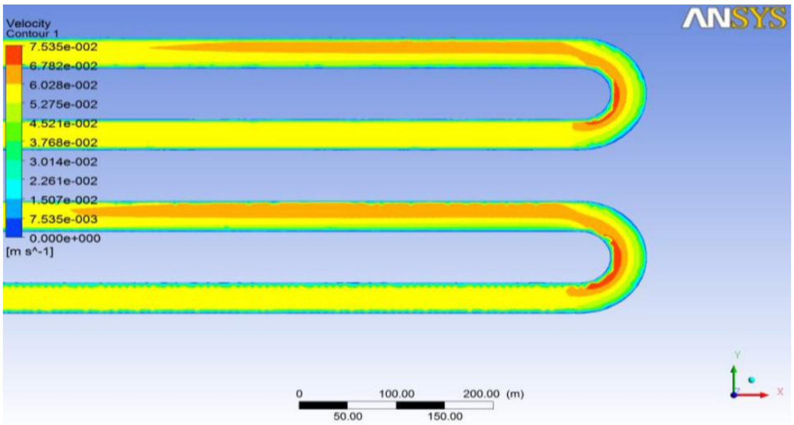

Fig. 9

VI. THERMAL ANALYSIS

Velocities and Pressure Profiles:

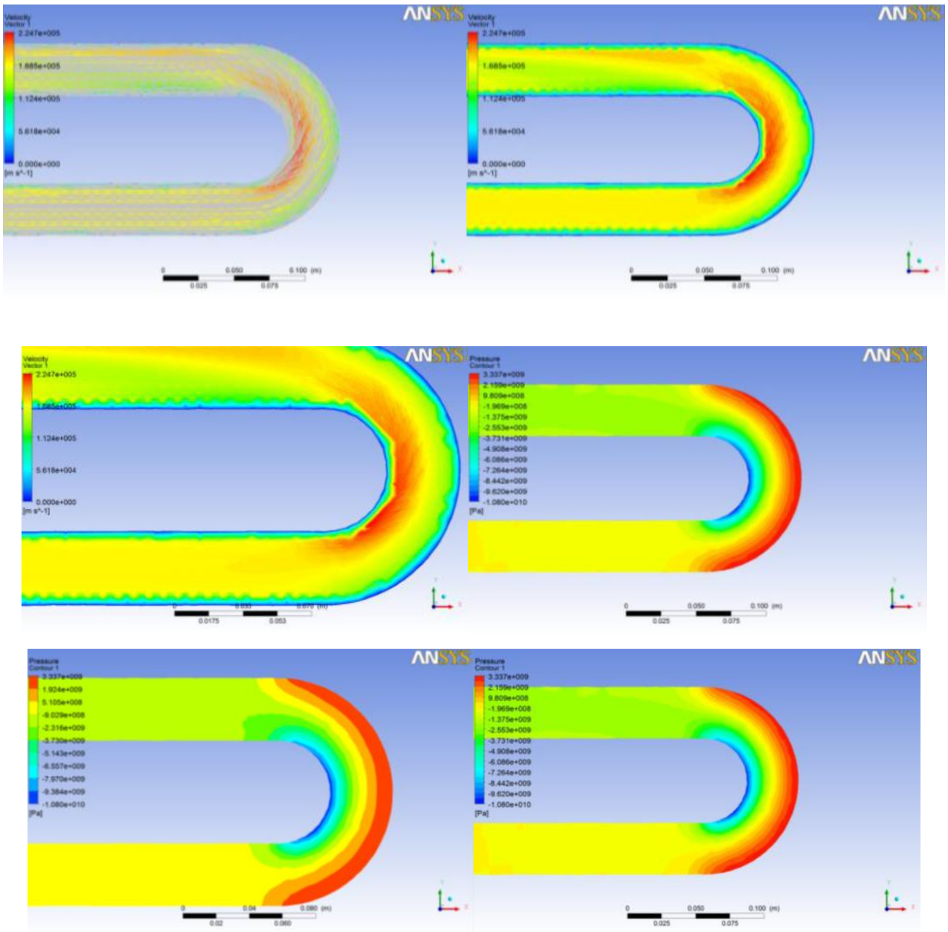

Fig. 10

Temperature Profiles:

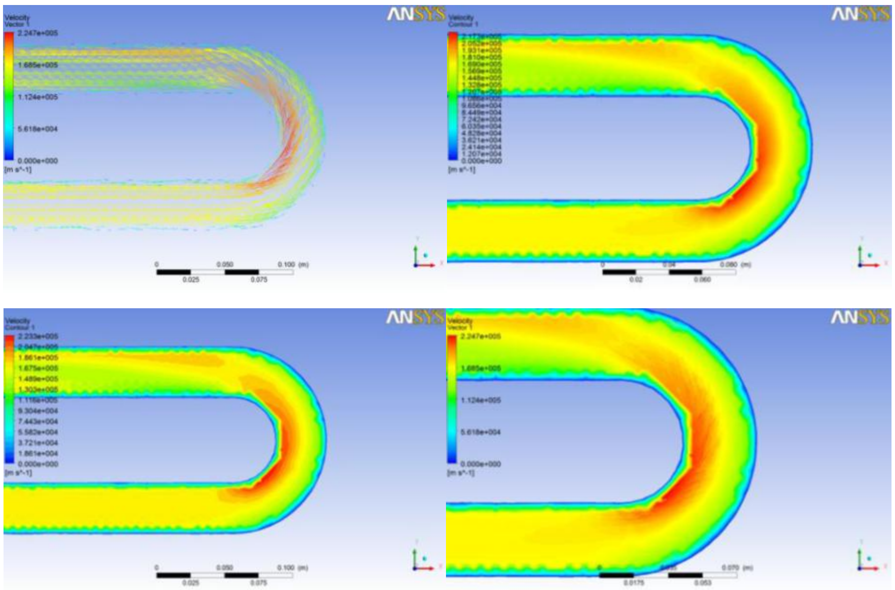

Fig. 11

VII. SUMMARY

From the obtained results, it is observed that the working fluid is accelerated at the C sections of the economizer tube. It was also observed from the obtained contour plots, the stream tube of the flow decreases in the C-section pipe, and resulting in the increase in velocity, which evident to the conservation of continuity equation. It has the maximum and minimum velocity magnitude as 7.535e-2 and 7.23e-3 respectively. Due to inertia of the fluid, to change its direction, the stream tube cross sectional area modifies to change its direction. Since considering working fluid as incompressible fluid, to compensate the change in cross sectional area, the fluid gets accelerate inside the stream tube, in the physic principle of mass conservation, which in other words, its satisfied the continuity equation. The pressure drop simulated in the economizer is that, it has the drop of 2.3 Pa, with reference pressure of 476000 Pa for every pipe. The overall pressure drop is measured 23 Pa across inlet and outlet.

REFRENCESS

[1.] Recovery boilers, American Forest & Paper Association, Tappi Press, Atlanta, 1997, ISBN 0-9625985-3.

[2.] Adams T.N., and Frederick W.J., Kraft recovery boiler physical and chemical processes.

[3.] American Paper Institute, New York, NY, 1988.

[4.] Backman R., Hupa M., and Hyöty P., Tappi J., 67 (12): 60-64, 1984.

[5.] Björklund H., Warnquist B., Pettersson B., Inside a Kraft recovery boiler – combustion of (high sulfidity)black liquor at high solids content, Tappi/CPPA 1989 International Chemical Recovery Conference Proceedings, Tappi Press, Atlanta, pp. 177- 82, 1989.

[6.] Blasiak W., Vaclavinek J., and Collin, R., Investigation of the flow conditions in a recovery boiler with respect to gases, droplets and dust, Project number: 1192, Department of Heat and Furnace Technology, Royal Institute of Technology, 1992.

[7.] Coulson J.M., and Richardson J.F., Chemical engineering, Volume 1; Fluid flow, heat transfer and mass transfer, 6th edition, The Bath Press, Bath, Great Britain, 2000.

[8.] FLUENT user guide, Fluentinc., 2001.

[9.] Grace T.M., Lien S., Schmidl W., Tse D., Abdullah Z., and Salcudean M., Validation of CFD-based recovery furnace models, International Chemical Recovery Conference, 1998.

[10.] Grimison E.D., Correlation and utilization of new data on flow resistance and heat transfer for cross flow of gases over tube banks, Trans. ASME, 59: 583-594, 1937.

[11.] Hupa M., Backman R., Skrifvars B-J., Tappi J., 73(6): 185-189, 1990.

[12.] Idelchik I.E., Handbook of hydraulic resistance, 3rd ed., Begell House, 1996, ISBN 1-56700-074-6.

[13.] Jacoby J., AF-Process AB, personal communication, 2005.

[14.] Jakob M., Heat transfer and flow resistance in cross flow of gases over tube banks, Trans ASME, 60: 384, 1938.

[15.] Kawaji M., Shen X.H., Tran H.N., Esaki S., and Dees C., Prediction of heat transfer in the Kraft recovery boiler superheater region, Tappi J., 78(10): 214-221, 1995.

[16.] Lundborg S., AF-Process AB, personal communication, 2005.

[17.] Patankar S.V., and Spalding, Heat exchangers: design and theory sourcebook, Scripta Book Co., ashington, D.C., pp.155-176, 1974.70

[18.] Salcudean M., Modelling of industrial processes using computational fluid dynamics, Canadian Metallurgical Quarterly, 37(3-4): 251-263, 1998.

[19.] Saviharju K., Pakarinen L., Wag K., and Välipakka I., Numerical modelling feedback in recovery boilers, International Chemical Recovery Conference, 2004.

[20.] Shen X.H., Kuhn D.C.S., Tran H.N., Mostaghimi, J., and Dees C., Simulation of flue gas flow in the upper furnace of a recovery boiler, Pulp & Paper Canada, 96(5): TI71-TI75, 1995.

[21.] Smook G.A., Handbook for pulp & paper technologists, 2nd edition, Angus Wilde Publications, 1992.

[22.] Tao L., and Blasiak W., Numerical simulation of a Kraft recovery boiler using rotation firing method, Modelling of Kraft recovery boilers - collection of publications (1), Division of Heatand Furnace Technology, Royal Institute of Technology, 1996, Tran H.N., Tappi J., 69 (11): 102, 1986.

[23.] Tse D., Matys P., Nowak P., Abdullah Z., Salcudean M., and Gartshore I., Flow and heat transfer modelling in the upper furnace of a Kraft recovery furnace, Department of Mechanical Engineering, University of British Columbia, Vancouver, B.C., Canada, 1996, in the 1998 final year report of Black liquor combustion validated recovery boiler modeling.

[24.] The Institute of Paper Science and Technology, Oregon State University, the University of British Columbia and Babcock & Wilcox Company.

[25.] Vakkilainen E.K., Adams T.N., and Horton R.R., The effect of recovery furnace bullnose designs on upper furnace flow and temperature profiles, Proc. of 1992 International Chemical Recovery Conference, J101-J112, Seattle, USA, 1992.

[26.] Vakkilainen E.K., Nikkanen S., Hautamaa J., and Anttonen T., Flows in the upper region of recovery boilers, AIChE Forest Products Symposium, 1991, pp. 125-134.

[27.] Versteeg H.K. &Malalasekera W., An introduction to computational fluid dynamics – the finite volume method, Addison Wesley Longman Limited, 1995, ISBN 0-582-21884-5.

[28]. Hema.K, Dr.S.Muralidharan. "NUCLEAR REACTOR CORE PRESSURE LOSS ADJUSTMENT WITH NEURAL APPROACH." *International Journal of Advanced Technology in Engineering and Science* 3.Special Issue No. 01 (2015): 49-57.