Experimental and computational fluid dynamics simulation of performance analysis of steam generators of boilers

Poor air temperature in modern power plants is one of the main reasons for the increase in unit temperature and the deterioration of boiler performance. The main problem of the air preheater is the leakage of air on the side of the gas flue and thus lead to improper thermal performance. The high ash content in Indian coal also adds to the problems associated with air tube heating. Air preheaters are designed to meet the operational requirements with consideration of the most impact parameters. heat transfer, leakage and pressure reduction. In the current work the performance of the air-conditioning heater is being tested with the help of the computational fluid dynamics (CFD) Fluent 14.0 analysis tool.

Keywords: Preheater, CFD, heat transfer coefficient, simulation technique.

I.0 Introduction

Hot air heating can increase boiler performance by about 1% at all 22°C at room temperature. The most common method of heating the air with a heat exchanger in the center of the flue. There are two types of air preheaters that can be used in electric generators for thermal power stations – one type is tubular built to boiler flue gas and the other is a renewable air boiler. These can be arranged so that the gas flows horizontally or horizontally across the rotational axis. Another type of air preheater is a regenerator used to make metal or glass. Many new circular beds (CFB) and bubbling fluidized bed (BFB) currently include tubular air heaters that provide advantages over moving parts of the circulation type [1-3]. With the rising cost of fuel and technology development, the size of the boiler that can be installed economically with a preheater should be smaller. While it is still the most effective technology for large boilers, high energy values will certainly motivate new ventilation systems before heating in very small boilers. Heaters can also use exhaust fumes or other energy sources depending on the system. The hot air produced by air heaters improves the heating of all fuels and is needed for drying and transporting fuel by coal-fired units. Industrial industries range from oil fires such as wood, sewage, industrial waste gases and coal, oil and natural gas. In small cone parts, plates and metal foil are widely used [6].

Air preheater is a common term to describe any design designed to heat the air before another process (e.g., combustion power) with the main purpose of improving the efficiency of the heating process. Air heaters can increase boiler performance and reduce chimney loss and standard power plant scheme with an air preheater as shown in Fig.1.1. It can be used alone or to replace the heating system or to replace the smoke coil. Air preheater (APH) is a tool used in boilers to move smoke from flue gases to the air of the fire before entering the furnace. The air heater recovers the heat from the flue gas boiler and increases the efficiency of the boiler. Flue gases sent to the gas tank at low temperatures allow the control of the temperature of the gases. Hot air preheaters used in large boilers found in thermal power plants generate electricity from that fuel, biomass or waste.

2.0 Objectives and methodology

To study computational fluid dynamics air preheater research on power plant using ANSYS fluent.

The purpose of the current study is to:

- Perform CFD measurements for air preheater to stabilize the outflow pattern.
- Perform a CFD simulation air preheater component to determine temperature and pressure output.
- Learn about the effect of installing a tube on air preheater to improve air temperature.

2.1 Methods and methodology to meet the objectives

- According to the literature review, the air preheater geometric model is created using CAD software such as design modeler
- Geometric decay and air preheater connection using ANSYS meshing processing tool.
- Study the effect filters and direct the vanes air preheater duct using ANSYS fluent
- Perform CFD simulation of air preheater to detect heat and pressure pressure using ANSYS Fluent.

Blind peer reviews carried out

Mr. Praveen Math, School of Mechanical Engineering, REVA University, Bangalore, Karnatka, India. e-mail: praveen6.math18@gmail.com

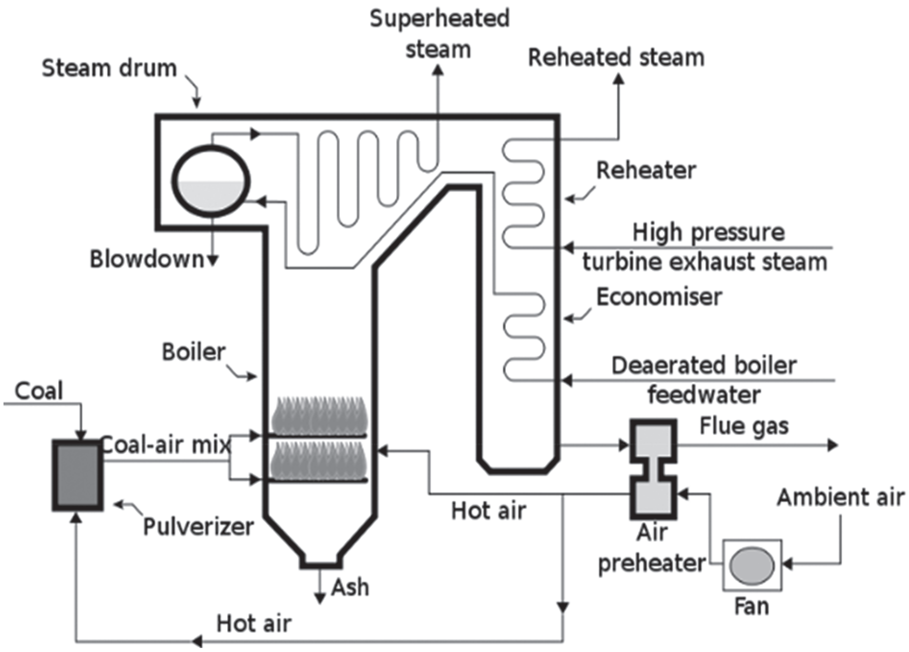

Fig.1: Schematic diagram of a steam generator in a conventional coal-fired power plant with air preheater

3.0 literatre review

USK Saha A. Dutta, [12] attempted to test the flow of servotherm oil in a circular acrylic tube lined with stainless steel that is made of solid. Learn about the effect of different lengths and widths of twisted tape with varying degrees of deviation in the heat transfer rate and the impact element. The main results were - Short length of twisted tape reduces pump loss but also reduces heat transfer, with the same tape high rate of heat transfer.

Zhi-Min Lin, Liang-Bi Wang, [13] in their experimental study of air flow in a circular Plexiglas tube used a curved metal bottom. Tapes with different twist ratios are used. They conclude that the tape increases the friction 3-4 times. With a small amount of twist, the maximum heat transfer is achieved compared to a large amount of twist.

2.2 Procedure Computational Method

The steps to finding the right liquid solution for Fluent are:

- Preliminary consideration: Including geometric construction, mesh formation in areas or volumes. This section is done with software ANSYS Meshing, connected to Fluent. Geometry was created in the ANSYS design modeler.
- Description of boundary conditions and other parameters, preliminary conditions, before starting to emulate in FLUENT, the match should be checked and measured and modified if necessary. Body types have to deal with it. These include the choice of pressure, viscosity, heat transfer considerations, laminar or turbulent flow, consistent or time-dependent flow. The boundary conditions should be clear because they specify the details of the flow situation in the specified areas: walls, scales, inlet air, exhaust air, etc.
- Troubleshooting, which is done by measurement until a combination of variables is found. First, the flow variables must be started and set to be calculated from a specific user-defined component. At this stage the flow balance is resolved. Pressure values are regularly updated and adjusted using repetition. The combination was tested to the maximum value set by the user.
- Post processing or result analysis is included. There are many options: Lines, XY sites, velocity vectors, path lines. In them, there are various analyzes: velocity, pressure, turbulence, strength, density and more

Watcharin Noothong et al. [14] their purpose is to investigate improvements in efficiency and to study the characteristics of heat transfer and thermal component conflicts. In the experimental study, a concentric double tube Plexiglas materialed heat exchanger was used. Cold water like annulus and hot air like indoor fluid used as a community. For internal tube stainless steel with no twist ratios was installed. They concluded with efficiency and the value of Nusselt increased by decreasing the twist rate and increasing the conflict by reducing the twist rate. Separation and blockage of the flow phase of the tube with tape, resulting in high flow rate. The second movement in the liquid is caused by a tape twist, and the resulting combination of the twist improves the convection heat transfer.

Paisarn Naphon, [15] in his experimental study used hot and cold water in a double-heated copper tube embedded in twisted aluminum tape. He studied the effects of temperature limits on heat transfer and pressure reduction. It has been concluded that the installation of twisted tapes has a significant impact on improving the rate of heat transfer. However, the pressure drop is also increasing. Adjustments to the equilibrium of heat transfer and friction factor according to test data are also presented.

Smith Eiamsa-ard et al., [16] their aim was to analyze heat transfer and the characteristics of a collision in a copper tube with a heat-resistant heat pipe, consisting of a stainless steel helical screw-tape or inside a rod inside. Hot and cold water are used for testing. They concluded that the installation of helical screw-tape has a significant impact on improving the rate of heat transfer and a significant increase in friction. Heat transfer rate from using helical screw-tape outside the core-rod is higher than that from a plain tube about 340%. The heat transfer rate obtained using tape outside the core-rod is found to be better than that made with a base rod around 25-60% while the friction is approximately 50% lower.

Ashis K. Mazumder and Sujoy K. Saha, [17] performed experimental studies on the screen and on the side of the acrylic pipes covered with full and short twisted tape. It was concluded that regularly spaced full length twisted tape performs better as compared to short length tape.

4.0 experiments and result

4.1 CFD analysis of air-preheater duct

In conventional tubular air heating, energy is transferred from the hot flue fluid that flows inside most enclosed tubes enclosed by the cold air of the fire flowing outside the tubes. The unit consists of a nest of straight tubes that are extended or inserted into tube sheets and placed in a metal case. In this case the straight type tubes are supported from the top or bottom sheet while the other (floating) sheet of the tube is free to move as the tubes extend inside the joint. Medium-sized plates and tube sheets are often used to separate the flow paths and to eliminate the flow that damages the tube. Carbon steel or rust-resistant materials are used in tubes ranging from 38 to 102 mm wide and with a wall thickness of 1.24 - up to 3.05 mm.

Larger, heavier gauge tubes are used where the power to insert the tube and rust is present. The most common order is the opposing flow and gas passing through tubes and air passing horizontally passing one or more without tubes. Variety of single-and-electric and air-conditioning arrangements used in plant formation. Designs often include provisions for the passage of cold air or the recurrence of hot air to control the final decomposition and cold ash

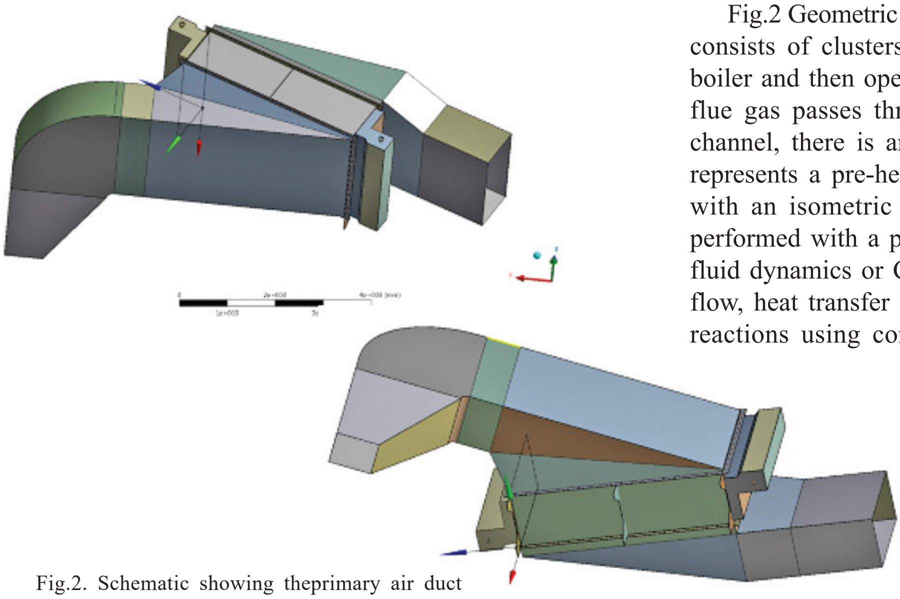

Fig.2. Schematic showing theprimary air duct

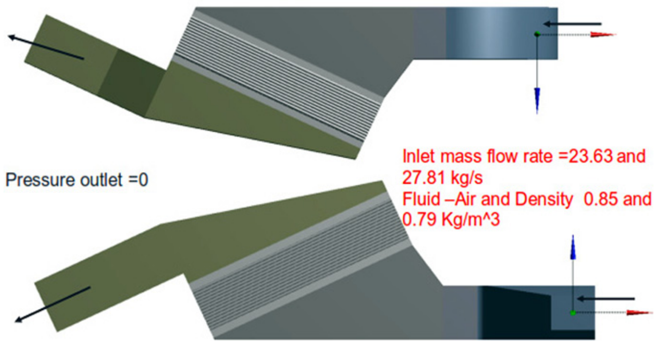

Fig.3: Boundary conditions

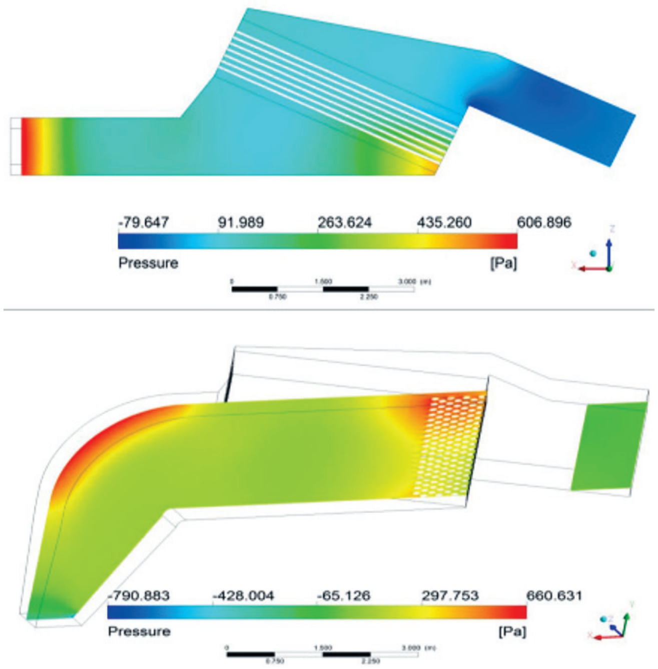

Fig.4. Pressure contours of air preheater duct

Fig.2 Geometric Model of air preheater, this tube air heater consists of clusters of straight pipes that run through the boiler and then open at each end without a hose. When the flue gas passes through the bending part of the flue gas channel, there is an imbalance in the flow patterns. Fig.2 represents a pre-heater module with an incoming gas pipe with an isometric view. Initially, CFD simulations were performed with a preheated heat exchanger. Computational fluid dynamics or CFD analysis systems that include water flow, heat transfer and related conditions such as chemical reactions using computer-based simulations. It works by resolving the flow of fluid in a region of interest in a defined condition at the boundary of that region.

The basic procedure for resolving any CFD problem is as follows:

1. The geometry (visible boundaries) of the problem is defined.

2. The fluid-filled volume is divided into separate cells (mesh).
3. Physical modeling is defined - for example, equality of movement + conservation of species
4. Defining boundary conditions. This includes specifying the performance of liquids and structures within the boundaries of a problem. For temporary problems, the initial conditions are also explained.
5. Statistics are extremely resolved as a static or a transitional case depending on the physical condition.
6. Analysis and evaluation of emerging solutions.

5.0 Results and discussions

Fig.4 shows the pressure pathways in the first duct path. As the fluid enters through the separating phase pipe and enters the elbow section, the pressure uneven with the flow of the vertical phase, must adjust the elbow to withstand centrifugal force. The pressure is very high on the outer wall very far from the bending center and at least on the inner wall adjacent to the bending center. In the elbow joint a low pressure is present on the inner wall and a high pressure is present on the outer wall, as clearly shown in Fig.3 Reduced base trench pressure is 927 Pa., 4.2 boundary conditions.

TABLE 1: FLOW BOUNDARY CONDITIONS

Parameter	Value
Mass flow inlet (kg/s)	23.63, 27.81
Pressure outlet	0

TABLE 2: INITIAL BOUNDARY CONDITIONS

Parameter	Value
Pressure	0
Velocity	0

This first gradient of pressure from shifting from right to curve, cross crossing force is located at the elbow, at the elbow port the outer layer of the outer wall gets the effect of a rough radio slope that is strong enough with the 900 elbow produces a local and vertical velocity as shown in the velocity this wall border shows the high turbulence created due to the opposite flow of the trench (Fig.7). Figs.7 and 8 show that recurrence occurs at the outlet of the elbow where the local pressure radiation arises from the flow of the stream accustomed to the same pressure conditions of the strem.

6.0 Conclusion

From the simulation results, the same flow of channel gas through the air preheater module can be seen as a result of adding a guide plate to the air preheater module entry points. Air preheater analysis was performed using the K model to predict air temperature in the air heating zone. Modeling of the pre-air vent geometry is done using the ANSYS design modeler and CFD meshing is done using the ANSYS meshing.

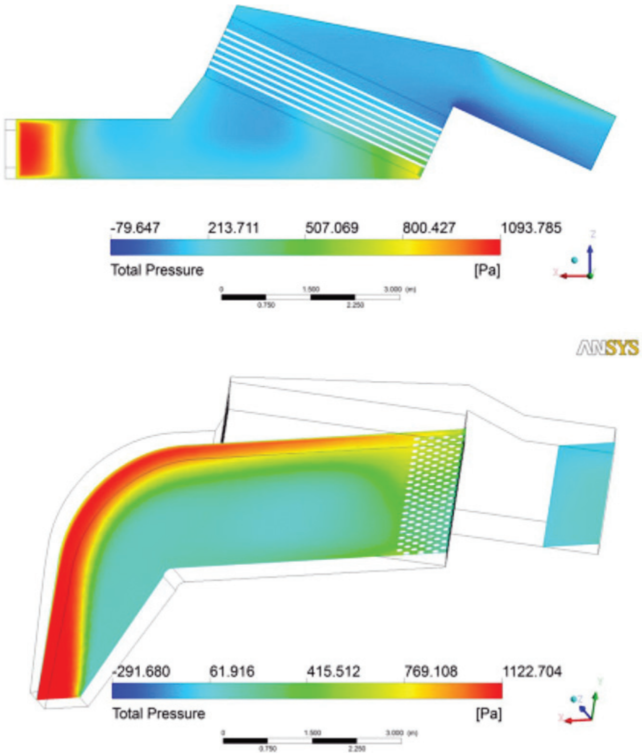

Fig.5: Total pressure contours of air preheater duct

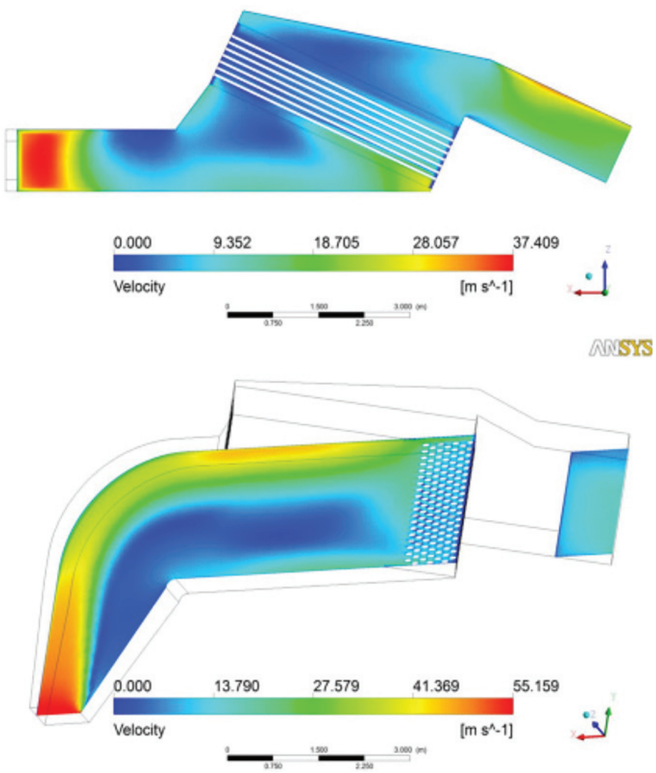

Fig.6: Velocity contours of air preheater duct

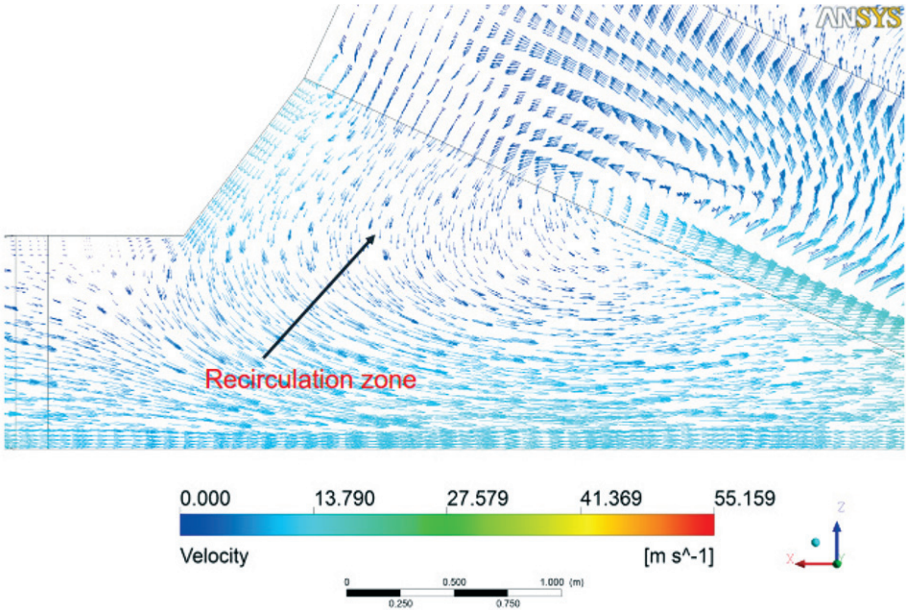

Fig.7: Velocity vector contours of air preheater duct

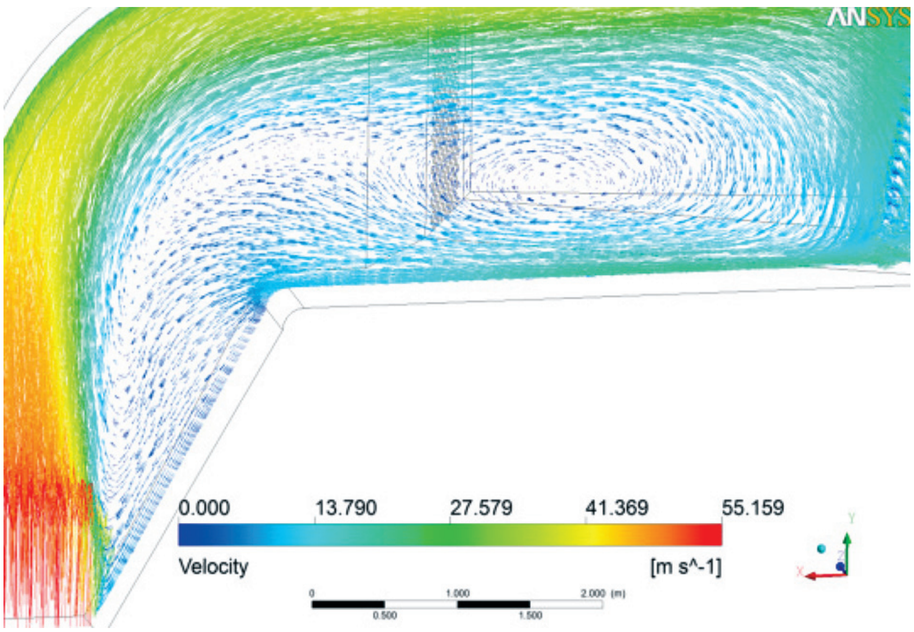

Fig.8: Velocity vector contours of air preheater duct-side view

Enhanced heat transfer coefficient of air preheater module results in reducing flue gas temperature at the outlet of air preheater. The modified design of the air preheater with tube installation has resulted in an increase of 17 to 20% of the temperature. CFD can be used as a tool to significantly reduce lead times and costs for new projects. The velocity velocity limits of the air preheater wind tunnel with detachable vans and steering vans that show the speed profile in the air-preheater space do not correspond to a separate phase because the flow tends to be one-sided and the vans angle needs to be further expanded to improve the flow pattern within the trench.

References

1. Staseik J.A. (1998): "Experimental studies of heat transfer and fluid flow across undulated heat exchanger surfaces", *Int. J. Heat Transfer.* Vol. 41 Nos. 6-7, pp. 899-914.
2. Skiepko. T. (1989): "Effect of reduction in seal clearances on leakages in a rotary heatexchanger", Heat recovery system CHP 9 (6), pp. 553-559.
3. Kumar, Rakesh, and Jain, Sanjeev, (2006): "Performance Evaluation of air pre heater at off designcondition", Dept of Mech. Engg., IIT, New Delhi, pp.1-4.
4. "Steam Book", (2006): The Babcock & Wilcox Company, pp.20-7.
5. Kem, Donald (2004): "Process Heat Transfer", Tata McGraw-Hill Publication, pp.701.
6. Skiepko, Teodor and Shah, Ramesh K. (1989): "A comparison of rotary regenerator theory and experimental results for an air preheater for a thermal power plant", Rochester Institute of Technology, Rochester, USA.
7. Skiepko, T. (1989): "Effect of reduction in seal clearances on leakages in a rotary heatexchanger", Heat recovery system CHP 9 (6), 553-559.
8. Kumar, Rakesh and Jain, Sanjeev (206): Performance Evaluation of air pre heater at off design condition", Dept of Mech Engg, IIT, New Delhi For books:
9. Gay, Rodney R. (204): "Power Plant Performance Monitoring", pp.433.
10. Saha, S.K.Saha and Dutta, A. (2001): "Thermo hydraulic study of laminar swirl flow through a circulartube fitted with twisted tapes" *Trans. ASME Journal of heat transfer*, June, Vol-123/ pages 417-427.
11. Lin, Zhi-Min and Wang, Liang-Bi (2009): "Convective heat transfer enhancement in a circular tubeusing twisted tape" *Trans ASME Journal of heat transfer*,

Aug ,Vol-131/081901-1-12.

12. Noothong, Wacharm, Eiamsa-ard, Smith and Promvonge Pongjet (206): "Effect of twisted tapeinserts on heat transfer in tube" 2nd joint international conference on "Sustainable Energy and Environment" Bangkok, Thiland.

13. Naphon, Paisam (2005): "Heat transfer and pressure drop in the horizontal double pipes with andwithout twisted tape insert" Elsevier Ltd.

14. Smith Eiamsa-ard, Smith, Thianpong, Chinaruk and Promvonge, Pongjet (2006): "Experimentalinvestigation of heat transfer and flow friction in a Circular tube fitted with regularly spaced twisted tape elements" *International Communications in Heat and Mass Transfer* Vol.33, Dec.

15. Mazumdar,Ashis K., Saha, Sujoy K. (2008): "Enhancement of Thermo hydraulic Performance ofTurbulent Flow in Rectangular and Square Ribbed Ducts With Twisted-Tape Inserts" *Journal of Heat Transfer August*, Vol.130.

16. A Dewan, A, Mahanta, P., Sumithra Raj, K. and Suresh Kumar, P.: "Review of passiveheat transfer augmentation techniques" Proc. Instn Mech. Engrs Vol. 218 Part A: J. Power and Energy, pages 509-527 S.S.Joshi et al. / *International Journal of Engineering Science and Technology* (IJEST) ISSN

17. Staseik J.A., (1998): "Experiment al studies of heat transfer and fluid flow across undulated heat exchanger surfaces", *Int. J. Heat Transfer*. Vol. 41 Nos. 6-7, pp. 899-914.

18. T. Skiepko, T. (1989): "Effect of reduction in seal clearances on leakages in a rotary heatexchanger", Heat recovery system CHP 9 (6), pp. 553-559.