International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 03 Issue: 11 | Nov -2016 www.irjet.net p-ISSN: 2395-0072

CFD MODELLING AND SIMULATION OF 500MW BISECTOR AIRPREHEATER AND ITS PERFORMANCE

K.VINOD BABA[1], P.POORNA MOHAN[2],Dr.T. JAYANANDA KUMAR[3]

[1]*Student, Thermal Engineering, Dept. of Mechanical Engineering, GIET(A), Rajahmundry*
[2]*Senior Assistant Professor, Dept. of Mechanical Engineering,GIET(A), Rajahmundry*
[3]*Professor, Dept. of Mechanical Engineering, GIET(A), Rajahmundry*

ABSTRACT*-Energy savings is one of the key issues in power generation systems, not only from the viewpoint of fuel consumption but also for the protection of global environment. In a power plant poor performance of air preheater is one of the main reasons for higher heat rate and is reponsible for lower efficiency of boiler. Performance of air preheater is significantly influenced by quantity of air leakage to flue gas side and its distribution. Air preheaters are aimed to meet performance chucks with consideration of greatly influencing factors viz. heat transfer, leakage and pressure drop.*

This project presents a Computational Fluid Dynamics Model for 500 MW bisector air preheater to study the individual seal leakages in the air preheater and their effect in the air preheater performance of thermal power plant. The geometry has been prepared using Gambit software. Flow and heat transfer have been modelled and analyzied using Fluent software. Standard K- ϵ model has been used for closure of the turbulence variable.Air preheater has been modelled and Simulated at the different loading conditions. The inlet temperature and pressure drop, have been compared with the actual data applied by the manufacturer. Effect of variation in the ambient temperature and seal clearance on leakage and its effect on air preheater performance have been reported.

Keywors- *Air preheater, lekage, efficiency, performance, cfd.*

1. INTRODUCTION

Rotary airpreheaters are used in all power station boiler units to recover heat from the flue gases. During operation the rotar of air preheater assumes the distorted shape with the casing and the non rotating seal surfaces remaning square, which would cause an increase in the gap at the hot end and decrease in the gap at the cold end. In an airpreheater usually the hot flue gas is under vaccum and the cold air is at high pressure. So, this high pressure air can leak to low pressure flue gas side through these gaps in hot state. Radial, bypass and axial seals are used in rotary airpreheater to minimize the leakage of high pressure cold air to low pressure hot flue gas side and also to prevent the flow bypass from inlet to outlet through the gap between the air preheater rotor and housing.These leakages due to the pressure difference are referred to as pressure leakages.

Also there is some unavoidable carryover of a small fraction of one fluid trapped in flow passages to other fluid stream due to the rotational effect of the rotor, and is referred to as carryover leakage. Numerical simulation of air preheater is an important tool to understand the fundamental mechanisms, flow pattern, various lekages and their distribution, as well as their effects on the heat transfer, pressure drop and air preheater effectiveness.

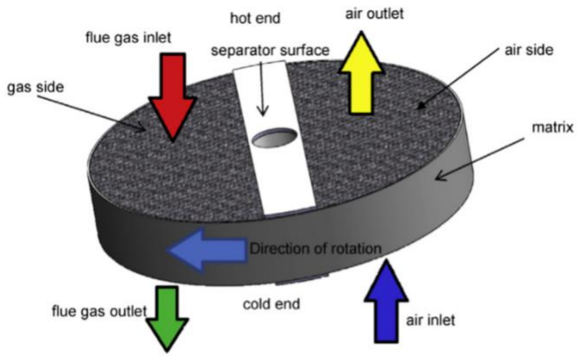

Figure 1.7 Schematic view of the Air preheater

2. LITERATURE REVIEW

Some of the previous numerical works on air preheater assumed the complete domain as a unit cell. Ciofalo.et.al[1] carried out an experimental and case study of flow and heat transfer for a crossed corrugated geometry, under transiant and weakly turbulent conditions. Three dimensional numerical predictions are obtained by a finite volume method using assumptions to standard and low Reynolds number K- ϵ turbulence models, direct simulation and large eddy simulation has been concluded by Skiepko[3] , the deterimental effect of axial seal lekages on the performance of air preheater.

The work of Teodor Skiepko, Ramesh K. Shah [2] on rotary air preheater of 5.3 m diameterresults demonstrating temperature distributions of heat exchanging gases and continuously rotating matrix are explained by means of 3D charts. The temperature distributions calculated are compared with experimental data. Right trends and a fair agreement between theory and experiments are found.The performance of tubular air preheater is assessed with the help of CFD analysis for staggered & in-line tube arrangementshas been studied byP.N.Sapkal [4]. The model can also be used while selecting a new type of surface geometry for optimising the design of air preheater.The directly leakage of rotary air preheater was studied by MingkumCai.[7] both experimentally and numerically tested different steel structures from single to triple seal cases which are used in really air preheater equipment to control the direct leakage.

The main factors that affect air leakage are the air flow expansion, inlet velocity at the seal gap entrance, and the flow boundary conditions on the seal plate surface. His studies concludes that the seal gap increases up to a critical value the difference between the double and triple seal becomes negligible.Armin heidari,Kaydan and Ebrahim H [8] are studied the thermal behaviour of full-scale rotary air preheater using the three-dimensional approach and treated preheater matrix has porous medium. The matrix porous medium and its mass momentum and energy equations are analyzed using moving reference frame approach to incorporate the effect of rotational speed of the matrix. The temperature distribution of the matrix at different conditions have been presented and the effect of essential parameters such as rotational speed of the matrix, fluid mass flow, matrix material and temperature of a inlet air on the performance of rotary airpreheater had been reported. They concluded that increasing the rotation speed of air preheater increases the efficiency upto certain limit and after that it does not considerable change and they has also found that the effect of material change is very limited on effeciency.

Performance of tubular air preheater is evaluated with the help of CFD analysis for In-line & staggered tube arrangement with the latter being more thermally efficientthe performance of tubular air preheater is evaluated with the help of CFD analysis for In-line & staggered tube arrangement with the latter being more thermally efficientthe performance of tubular air preheater is evaluated with the help of CFD analysis for In-line & staggered tube arrangement with the latter being more thermally efficientthe performance of tubular air preheater is evaluated with the help of CFD analysis for In-line & staggered tube arrangement with the latter being more thermally efficient.

In the present work entire air preheater has been modelled as the computaional domain. For the selected geometry, three dimensional unsteady state turbulent simulations has been carried using a finite volume based CFD package. Gambit was used as the preprocessor for geometry creation and the meshes generation by FLUENT was used as the solver and post processor. Inputs for the simulator are inlet pressure, inlet temperature, turbulent parameters at inlet for air and flue gas, composition of flue gas at inlet. The details of the simulation and its results are presented in the following sections.

3. GEOMETRICAL AND MATHEMATICAL MODELLING

3.1 Geometry creation

The computational domain was created in form of three tiers of cylindrical volume. The top volume has been spilt by a plane, having the form of lateral surface of the frustum of cone such that its height from the bottom plane of the top cylinderical volume at inboard towards the rotor inner periphery and the outboard towards the rotor outer periphery are equivalent to the hot and radial seal clearance in hot end resulting in two volumes in top of cylindrical volume. Upper volume of the two has been split without connection, into two angular segments by means of radial planes, representing flow field of flue gas and air. Lower volume represent the hot end radial seal zone. Similarly, the bottom volume have been split by a plane, during the form of lateral surface of a various approaches ranging from laminar flow frustum of cone such that its high height from the top plane of the bottom cylindrical volume at inboard and outboard are equivalent to the cold end radial seal clearance in hot state, resulting in two volumes in bottom tier of cylindrical volume. Lower volume of the two has been split without connection, into two angular segments by means of radial planes, representing flow field of flue gas and air. Upper volume represent the cold end radial seal leakage zone.

The rotor bypass zone has been modelled as annular space between the lateral surface of a frustum of cone outside the rotor, coaxial with the rotor axis, and the rotor outer surface such that the gap between the frustum of cone and cylindrical rotor are equal to hot state and cold end of air preheater. This rotor bypass zone has been spilt without connection into two angular segments, through out the rotar height, by means of radial planes aligned with top and bottom volume radial planes.Bottom surface of lower volume in top tier above rotor and top surface of upper volume in the bottom tier below rotor have been split into two surfaces, one having inner and outer diameter equal to that of rotor and other having inner diameter equal to that of frustum of same diameter at hot and cold end.

In order to enable the flow of flue gas and air through air preheater, the top and bottom surfaces of the rotor are defined as interface with inner bottom surface of top volume and inner top surface of bottom volume respectively.Similarly for bypass flow, top and bottom surfaces of the rotor bypass zone were defined as interface with outer bottom and top surfaces of top volumes above rotor and bottom volumes below rotors respectively. Inner and Outer cylindrical surfaces as well as the radial dividing planes above and below the air preheater rotor and in bypass zone were defined as wall.

Isometric meshed view of the modelled Air preheater is as shown in figure below.

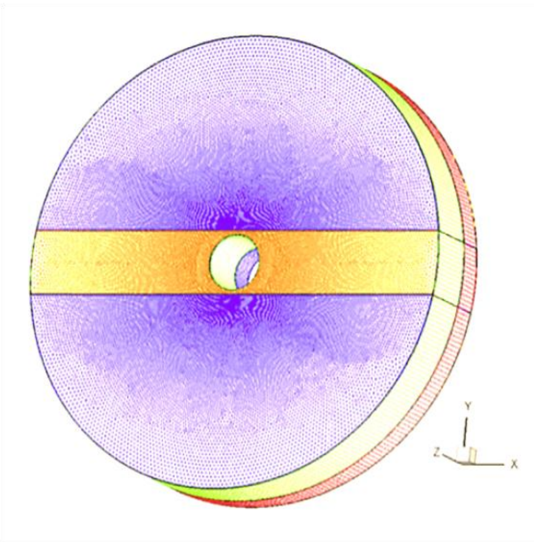

Figure 3.1 Rotary air pre heater meshed grid view.

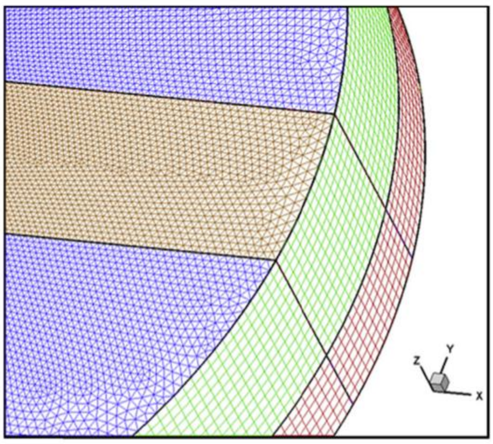

Figure 3.2 Closed view of meshed grids

The rotor of the air preheater has been modelled as intermediate cylindrical volume. This intermediate volume is a porous medium, having porosity only parallel to the rotor in the direction of z- axis and no porosity in the other directions.

3.2 Mathematicalmodelling

The simulation of air preheater performance involves the modeling of turbulent flow and heat transfer between the gas and rotor & between rotor and unsteady one dimensional simulation of the air preheater has been carried out. The following exemptions have been taken for the simulation process.

1) Air preheater rotor consisting of very large number of corrugated metallic sheets working as the heat transferring elements have been modelled as a porous medium.

2) The flue gas has been considered as a single phase gaseous mixture free from the ash.

3.2.1 Governing Equqtions

The mathematical modelling of the system involves the choice of the governing equations and discretisation scheme.

Mass Momentum and energy equations can be written as

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho V) = 0 \qquad (1)$$

$$\frac{\partial(\rho V)}{\partial t} + \nabla.(\rho V.V) = -\nabla p + \nabla\tau + S_i \qquad (2)$$

Porous media is model by the addition of a momentum sink term to the fluid flow equations. The sink term is composed of two parts of viscous loss terms, intertial loss term as follows.

$$S_i = -\left(\frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho v_{avg} v\right) \qquad (3)$$

$$\frac{\partial(\gamma\rho\vec{v})}{\partial t} + \nabla\cdot(\gamma\rho\vec{v}\vec{v}) = -\gamma\nabla p + \nabla\cdot(\gamma\vec{\tau}) + \gamma\vec{B}_f - \left(\frac{\mu}{\alpha} + \frac{C_2\rho}{2}|\vec{v}|\right)\vec{v} \qquad (4)$$

Transport equations for the standard k- ϵ model

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k \qquad (5)$$

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(G_k + C_{3\epsilon}G_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon \qquad (6)$$

In these equations, $G_k$represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. σ_k and σ_ϵ are the turbulent Prandtl numbers for k and ϵ, respectively. S_k and S_ϵ are user-defined source terms.

The spices transport equation is

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla.(\rho v Y_i) = -\nabla.J_i + S_i \quad (7)$$

The Dimentional data and operational parameters of the rotary air preheater is as shown in table below.

S.No	Parameter	Values
1	Boiler Size (electrical output)	500MW
2	Fuel	Coal
3	Ljungstrom Air Preheater Size	2 x 30
4	Air preheater diameter (m)	10.6
5	Area (m^2)	88.20
6	Rotation Speed (rpm)	2
7	Gas flow (Nm^3/h)	610 000
8	Air flow (Nm^3/h)	575 000
9	Gas temperature inlet (°C)	380
10	Gas temperature outlet (°C)	130
11	Air temperature inlet (°C)	29
12	Air temperature outlet(°C)	320
13	Inlet air pressure at outlet (KPa)	2.88
14	Inlet air pressure at outlet (Kpa)	1.75

Table 1: Technical parameters of Air preheaters.

The above governing equations and technical data has been used for modelling and simulation of air preheater in FLUENT software by applying the boundary conditions and results are compared with the available plant data. The variations are reported in later sections.

4. BOUNDARY CONDITIONS AND SOLUTION METHOD

4.1.1 Inlet boundary conditions

Pressure inlet boundary condition have been used to define the fluid pressure at the flow inlet along with total temperature for both flue gas and air. Pressure as assumed uniform over the entire inlet plane for both air and flue gas. Turbulence intensity and hydraulic diameter are also specified for solving turbulence equations. For internal flow the value of turbulence intensity at the inlet depends totally on the upstream history of flow.

4.1.2 Outlet boundary conditions

Pressure outlet boundary condition have been used to define the fluid pressure at the flow outlet along with temperature of reverse flow for both flue gas and air. As actual pressure at outlet for both fluid streams are not known but the mass flow rates at inlet are known. Outflow boundary condition has not been used in this model due to the relative nature of pressure distribution in flow field, where as the leakage depends on the actual present difference between the fluids

4.1.3 Wall boundary conditions

All the inner and outer circumferential walls and the radial dividing walls above and below the porous media have been specified as insulated and stationary walls of steel. Inner and outer circumferential walls of porous media have also been specified as insulated walls of steel but having same rotational speed has that of the porous medium.

4.1.4 Porous media conditions

The porous media consists of enameled steal as solid. So a fixed value of porosity of porous media has been specified and the properties of steel are assigned to the solid in porous medium from FLUENT database. Ergum equation has been used for calculating viscous and inertial resistance coefficients in the flow direction (z-axis), as pressure drop and velocity of fluids are known. As there is no flow in other two directions viscous and inertial resistance coefficient in radial and angular directions have been specified thousand times greater than in flow direction (z- axis).

Moving reference frame have been used to specify the rotational motion to the porous medium. Speed of rotation of porous media is 1.04 rpm, with z-axis as the axis of rotation.

4.2 Solution Methodology

The unsteady state governing equations described in chapter 4 have been followed by using segregated solver with implicit formulation having a convergence criterion of 10 to 15 for each time step. For each time step maximum number of iteration have been selected has hundred. Iterations were continued until either of the above mentioned two conditions are satisfied. Time step size has been selected as one and two seconds. This process was repeated until the mass and energy balance was achieved.

Due to large computational domain and more number of equations computation time was enormous. Major difficulty was encountered during modelling, due to the dimension of air preheater rotor and very small clearances. Meshing of the very small lekage control volumes was a main challenge.

5.RESULTS AND DISCUSSION

The model of air preheater have been simulated at different loading conditions of BMCR and the simulated value of pressure drop for gas and air are 6.1 % and 1.3% more than the rated values respectively. Where as the mass flow rate of gas and air are 0.73 and 0.27 % less than the rated value respectively. Simulated temperature value of air preheater outlet for gases 15.9 % higher than the rated value where as for air it is 19.1% lesser than the rated value.

At TMCR, simulated value of pressure drop for gas is 3.3 % higherand that of air and 0.8 % less than the rated value, changed until the simulated mass flow rate matches with the actual mass flow rate for both fluid streams values. The simulated value of temerature at air preheater outlet for gas is 14.4% higher than the rated value, where as for air it is 19.4 % less than the rated value. The difference in temperature may be due to the assumption of rotor as porous medium having the porosity only in flow direction, so that the fluids are passing through the rotor smothly without any additional turbulence and the assumption of thermal equilibrium between the medium and the fluid flow in FLUENT. Quantity of air leakage is the summation of leakages through all seals.

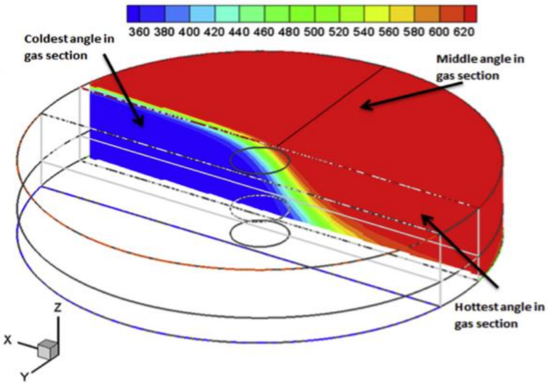

Figure 5.1 Temperture distribution inside the air preheater

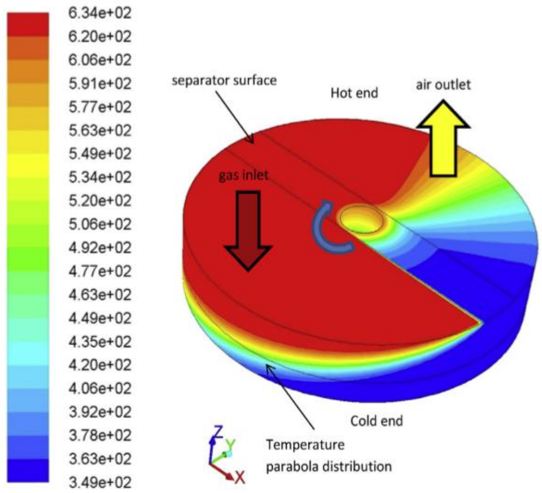

Figure 5.2 Temperture distribution of the air preheater when speed is 2 rpm

The flue gas and air enters the rotor of air preheater through the stationary volume from top and bottom respectively, the path lines are bending due to the rotational effect of rotor. In the rotor, even though pathlines are straight but inclined. Similarly as gas and air exits the rotor of air preheater, to the stationary volume from the bottom and top respectively,the path lines are bending values respectively.

Due to decrease in gas temperature in flow direction, its velocity is decreasing. Whereas for air, velocity is increasing due to increase in its temperature.There is no significant temperature gradiant in circumfencial direction over the entire surface of the air preheater rotor. This is due to the assumption of thermal equilibrium between the medium and the fluid flow, in Fluent. In Fluent standard energy equation is solved for the porous region with only modification in the conduction flux and transiant term. Conduction flux terms uses effective thermal conductivity, which is the volume rated average of both solid and fluid phase between the transient trem includes the thermal inertia of the solid region on the medium. This is the reason for difference in rated and simulated value of air and flue gas outlet temperature.

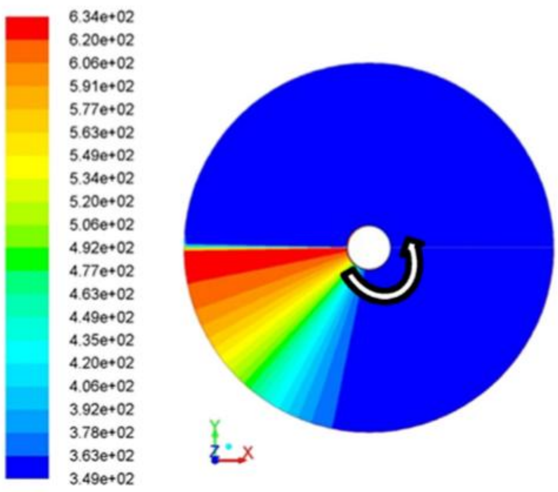

Figure 5.3 Temperature countour at bottom of air preheater

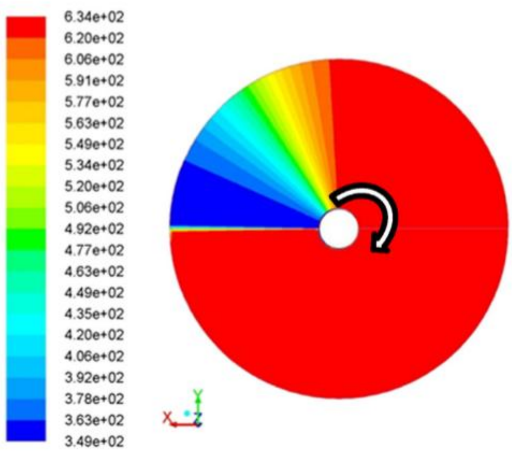

Figure 5.4 Temperature countour at top of air preheater

5.1 Effect of variation in air inlet temperature on air preheater Performance

In order to study the effect of variation in air inlet temperature (ambient temperature) on air preheater performance, air inlet temperature was changed from 0°C to 50°C whereas the gas inlet temperature was kept constant.The air inlet temperature increases, air and gas outlet temperatures are also increasing due to increase in air inlet temperature from 0°C to 50°C, change in air and gas temperature from inlet to outlet decrease by 27.1 °C and 24 °C. The simulated values are tabulated shown in following tables.With increasing air inlet temperature, gas and air side pressure drop is also increasing 44.8 Pa and 43.4 Pa respectively. The results of the simulation at full load are plotted in figure 5.5 and figure 5.6.

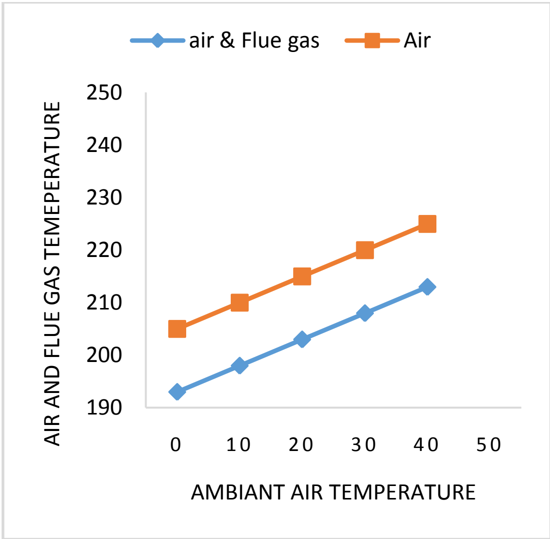

Figure 5.5 Effect of variation in ambient temperature of air on air and flue gas outlet temperature

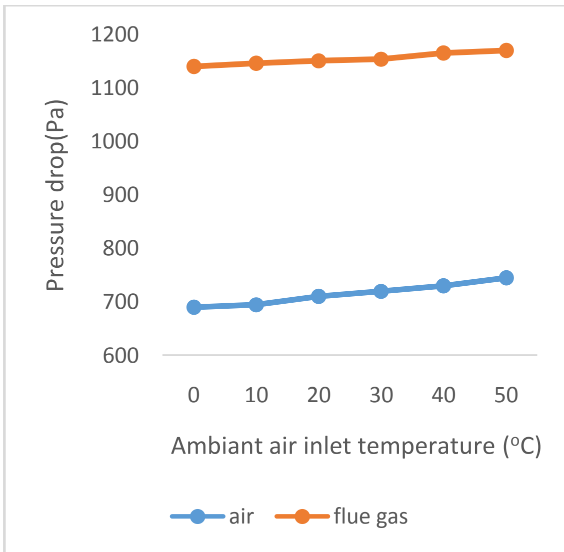

Figure 5.6 Effect of variation in ambient temperature of air on air and flue gas Pressure drop

5.2 Air preheater with variable hot and cold and radial seal clearance

In an air preheater various seals wear out due to flow of high velocity dust laden flue gas and air in due course of time. This wear out is more prominent at the hot end, where the high velocity gas enters the air preheater. The wearing out of seals depends upon the flow pattern of flue gas and air. As a result the seal clearance goes on increasing with operating time.In order to study the effect of variation in hot end radial seal clearance on leakage and air preheater performance, simulations have been carried out by keeping hot end outboard radial seal clearance fixed at 22 mm and 50 mm while inboard radial seal clearance have been increased gradually, keeping other seal clearances fixed at initial value. The performance parameter of air preheater have been plotted against percentage seal leakage in figure 5.7 and 5.8.

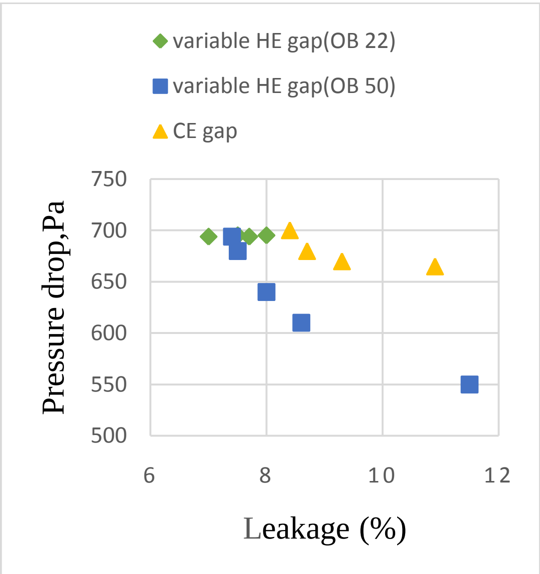

Figure 5.7 Variation of air pressure drop with leakage

At hot end for both outboard radial sale clearances, as inboard radial seal clearance increases due to increase in seal clearance area, leakage increases and air pressure drop reduces. With increase in outboard radial seal clearance increase in air pressure drop has been observed. While with increase in cold end radial seal clearance, air pressure drop is continuously decreasing, due to decreasing leakage at cold end and decrease in mass flow rate through rotor.

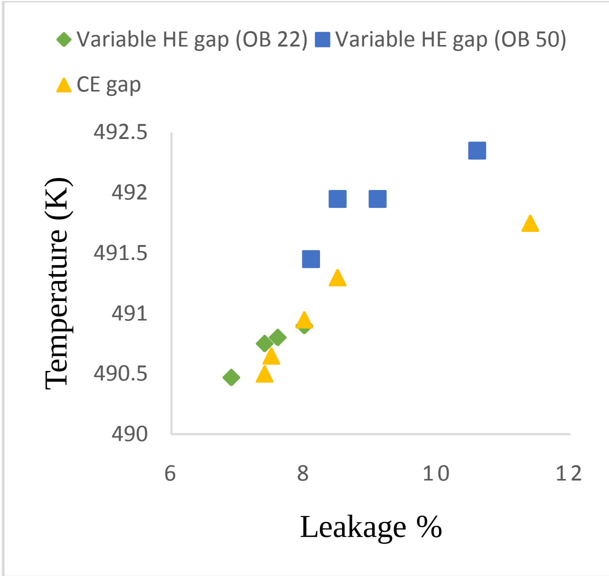

Figure 5.8 Variation of air outlet temperature with leakage

In figure 5.8with increase in leakage at hot end, hot air leaks to gas stream resulting in more energy available for transfer to the cold air. While increase in leakage at outer end causes reduction in mass flow rate of air through the rotor. Due to reduced flow rate of air, its outlet temperature increases.

Increase in hot end radial seal leakage is more severe compared to increase at outer end radial seal lekage, as it increases the air pressure drop and hence increases the fan power consumption. The effect of increase in radial seal leakage has negligible effect on actual heat transfer to cold fluid. These results are similar to that ofshah and skiepo [6].

6. CONCLUSIONS

The modelling and simulation of a power plant air preheater has been carried out at different loading conditions, to protect seal leakages and their effect on air preheater performance. The complete unit have been taken as the computational domain and the rotary assumed to be porous media. The results obtained are analyzed and validated with rated values.

- Results of simulation for air preheater shows that air and gas side pressure drop are in good quantitative agreement, while the air and gas outlet temperatures are within 14 - 20% with the rated value.
- These differences in gas and air outlet temperature seems to be due to limitation of porous media modelling in FLUENT.

- The effect of ambient air temperature and seal clearances on the air preheater performance has been also been reported.
- At BMCR increase in ambient temperature from 0°C to 50°C increases the pressure drop for air and gas by 43.4 Pa and 44.9 Pa respectively, and the gas outlet temperature by 24 °C. Hence with increase in ambient temperature, both fan consumption and boiler loss increases.
- At BMCR with all combinations of seal clearances it had been found that with increase in seal leakage both boiler losses and pressure drop increases. Thus increased in seal leakage is decreasing the power plant efficiency. Amongst the radial seals, hot end radial seal leakage adversely affects the power plant efficiency.

NOMENCLATURE

TMCR	Turbine maximum continous rating
BMCR	Boiler maximum continous rating
PTFE	Poly TetraFuoroethylene
G_k	Generation of turbulence kinetic energy due to the mean velocity gradients
G_b	Generation of turbulence kinetic energy due to buoyancy
Y_M	Fluctuating dilatation in compressible turbulence to the overall dissipation rate.
σ_k , σ_ϵ	Turbulent Prandtl numbers for k and ϵ
S_k ,S_ϵ	Defined source terms
C_2	Inertial resistance factor
k	Turbulent kinetic energy
k_{eff}	Effective thermal conductivity of the medium
IB	In Board
OB	Out Board
k_f	Fluid phase thermal conductivity
k_s	Solid medium conductivity
Re	Reynolds number
ASME	American Society of Mechanical Engineers
TEMA	TubularExchanger Manufacturers Association

REFERENCES

[1]. Michele Ciofalo, 1998, A computational approach to conjugate heat transfer between two fluids in plate heat exchangers of arbitrary geometry.

[2]. Teodor Skiepko, Ramesh K. Shah , "A comparison of rotary regenerator theory and experimental results for an air preheater for a thermal power plant", Rochester Institute of Technology, Rochester,USA.

[3]. T. Skiepko, "Effect of reduction in seal clearances on leakages in a rotary heat exchanger", Heat recovery system CHP 9 (6) (1989) 553-559.

[4]. P.N.Sapkal, P.R.Baviskar, M.J.Sable, S.B.Barve "To optimise air preheater design for better performance".

[5]. TeodorSkiepko&Ramesh K. Shah "Modelling and effect of leakages on heat transfer performance of fixed matrix regenerators".

[6]. TeodorSkiepko&Ramesh K. Shah "Influence of leakage distribution on the thermal performance of rotary regenerator" Applied thermal engineering.vol.19, 1999.

[7].MingkumCai, SheinHui, Xuebin Wang "A study on the direct leakage of rotary air preheater with multiple fields".

[8] Armin heidari,Kaydan and Ebrahim H three dimensional simulation of rotary air pre heater in steam power plant Elsevier 2014.

[9]. Mr. Vishwanath .H. H, Dr. Thammaiah Gowda , Mr. Ravi S.D " Preheater" International Journal of Innovative Research in Science, Engineering and Technology July 2013.

[10]. Staseik J.A., "Experimental studies of heat transfer and fluid flow across undulated heat exchanger surfaces", J. Heat Transfer. Vol. 41 Nos. 6-7,pp. 899.

[11]. Larsen F. W., "Rapid Calculation of Temperature in a Regenerative Heat Exchanger Having Arbitrary Initial Solid and Entering Fluid Temperatures.

[12]. Wang .H," Analysis on Thermal Stress Deformation of J. Chem. Eng., Vol. 26, 833-839 , 2009. [6]. T.Skiepko, "Effect Of Reduction In Seal Clearances On Leakages In A Rotary Heat Exchanger system CHP 9 (6), pp. 553-559, 1989.

[13]. Stephen.Storm, john, Guffre, Andrea Zucchelli " Performance And Reliability" POWERGEN Europe.