Numerical analysis and experimental investigation in a heat recovery generator

[#1]Dattatraya Kad, [#2]Prof. Shriramshastri Chavali, [#3]Prof. Vivekanand Navadagi

[1]PG Student
[2]Associate Professor, Department of Mechanical Engineering, DPCOE Wagholi Pune
[3]Assistant Professor, Department of Mechanical Engineering, DPCOE Wagholi Pune

Abstract: Boiler is a metal container in which a fluid is heated upto it changes to vapor or high temperature liquid. Heat supplied in boilers by means of convection, radiation and conduction. There are two type's boiler fire tube boilers and water tube boilers. Here we analyzed water tube boiler. It is very essential to optimize and study the heat transferring mechanism and their characteristics to study & also control the various thermal losses. In this work internal flow analysis of a super heater is done to study the heat transfer characteristics of super heater using a Computational Fluid Dynamics (ANSYS- FLUENT) package. The CFD analysis provided fluid velocity, temperature, pressure, wall fluxes and especially we have concentrate from inlet to outlet of the super heater of a boiler pressure drop. The Computational Fluid Dynamics (CFD) approach used here to solve many boiler problems such as pressure drop, parametric study and heat losses of super-heater analysis helped to study the possibilities of improve the heat transfer properties.

Keywords: **FD, Wall fluxes. Introduction**

I. INTRODUCTION

The boiler is the device of power plant which generates steam by burning of available fuels used to generation of power. The super heater can be works as the heart of any boiler system. The main work of which is to supply desired amount of steam continuously at rated pressure and temperature. Super-heater is the heat exchanger in which heat is transferred from furnace gas to the steam, Because of the improper heat transfer between furnace gas & steam leads to problems of heating. It reduced performance, repetitive failures in boiler component are common problem related to any type of boiler system. Super heater tube damage is very common issue in boilers. In this work we have done CFD analysis of super heater flow to study thermal parameter, to study and investigate the enhancement in heat transfer characteristics. Temperature distribution in the water tube boiler performs various efficiency testing and simulation of thermal flow inside in sugar factory boiler, The analysis of the temperature distribution for any location inside the domain is conducted by set's the constant temperature and varying parameters such as mass flow rate of steam (kg/s), steam inlet temperature and depth of scale formation. The commercial CFD software used to control volume based technique to convert the various governing equations, which are solved numerically using the implicit method. The temperature distribution in the boiler tube is affected by the many variables such as mass flow rate of steam (kg/s), steam temperature, feed water pressure & temperature. If we increase the mass flow rate of steam through the boiler tube then decrease in temperature in the inner tube wall. Computer simulation is used to understand the thermal flow in the boiler to solve the operational problem and search for optimal solution. The thermal flow behaves inside the boiler was studied to make the study of heat transfer characteristics and minimize the all thermal losses. The work study performs a detailed simulation of combustion and thermal flow behaviour inside the sugar factory boiler. Actual Working Conditions

Fig.1.1 Actual photograph of Steam drums of a super heater

© 2015, IERJ All Rights Reserved

Fig.1.2 Actual photograph Super heater tubes of a boiler

The existing super heater of a boiler is manufactured by Walchandnagar Industries Ltd. and assembled by Hi-Tech Engineering Corporations for the Shrinath Mhaskoba Sakhar karkhana Pvt. Ltd. Patethan Tal: Daund, Dist:Pune-412104. This plant has capacity of 70 Tons/hr. In this sugar factory there are two types of superheating coils primary coils and secondary coils. Heated water and steam from the heating pipe is enters in the steam drum as shown in fig 1 and there is separator in the steam drum so that water and steam is separated, below 300 °c steam is reheat and the only above 300°c steam flows in the super heater pipe this steam temperature again increased by due to heating of flue gases in the boiler drum .In sugar factory there are total 45 number of same super heating tubes present at the separator drum arrangement. The capacity is 70 Tons/hr steam is flow through the super heater tubes. Same steam flow rate gets divided in 45 numbers of tubes, so we have consider single tube for the analysis.

1.2. Objectives of Work

The objectives of work are as listed below.

1. To study the velocity, pressure and temperature distribution in the super heater that assists to find causes of heat losses and identify the decreased heat transfer characteristics through CFD simulations.
2. To validate the practical result with CFD result.
3. To find feasible and practically usable solution to enhancement in heat transfer characteristics in the super heater by changing the different parameter in the super heater like geometry of the system, boundary conditions such as mass flow rate, diameter of coil, inlet temperature to view the thermal flow etc. by parametric study.

II. LITERATURE REVIEW

The aim of the present work is provide the detailed review on problems with super heater tube damage and enhancement proposed by the various authors to improve the heat transfer characteristics. The losses accounted and measured are also reported. The present will help to investigate the various aspects of heat transfer and fluid flow in the super-heater section. Therefore this chapter concentrates on the modeled and analyzed super heater to study and understand the thermal flow in the boiler to resolve the operational problems.

Saripalli and Masoud [1-2] presented the detailed thermal-flow and combustion in the boiler and showing possible reasons for super heater tube rupture. The exhaust gas temperature is consistent with the actual results from the infrared thermograph inspection. It helps industry to improve boilers efficiency, reduce emissions, to avoid rupture of super heater tubes and also to understand the thermal flow transport in the boiler. The Computational Fluid Dynamics (CFD) scheme applied for to show an overall picture of what is happening within the boiler, making it easy in most cases to identify the problems and develop a solution. A Computational Fluid Dynamics (CFD) analysis provides fluid velocity, pressure, temperature, and species concentrations throughout the entire solution domain. During the analysis the boundary conditions such as flow rate and inlet velocity can be easily changed to view their effects on thermal-flow patterns or species concentration distributions. Several ideas were formed from this study to improve boiler efficiency and minimize the thermal stress problem imposed on the super-heater tubes.

Jayakumar[3] reported that heat transfer in a helical coil is higher than that in a corresponding straight pipe. It was observed that the variation of local Nusselt number along the length and circumference at the wall of a helical pipe. Movement of fluid particle in the helical pipe has been observed. CFD simulations are carried out for vertically oriented helical coil by varying coil parameters such as (i) pitch circle diameter (PCD), (ii) tube pitch & (iii) pipe diameter and their influence on heat transfer has study. Many researchers have identified that a complex flow pattern exist inside a helical pipe due to which the enhancement in heat transfer is find out. Transition from laminar to turbulent flow takes place at a Reynolds number higher than that for a straight pipe. Critical Reynolds Number and curvature ratio given bellow. A plot of 〖Re〗cr for the curvature ratio from 0.01 to 0.25 is given in Fig.2.1 In the lower range of curvature ratios (1 < 0.05), all of the correlations provide approximately the same value for 〖Re〗cr. The periodic behavior of Nusselt number at top and bottom sides of the cross-sections along the length of the pipe.The values of Nusselt number at the inner, outer, top and bottom along the periphery at given cross- section in the developing section. Fig.2.2 gives the variation of Nusselt number around the periphery at various cross-sections, along the length of the helical pipe. In this analysis, a helical coil with a pipe of inner diameter (2r) 20mmand pitch circle diameter of 300mm was considered. Analyses were carried out by changing the coil pitch. Coil with pitch of (i) 0mm, (ii) 15mm, (iii) 30mm, (iv) 45mm and (v) 60mm were analyzed. When the coil pitch is zero, local Nusselt numbers at the bottom and top points on cross of the

© 2015, IERJ All Rights Reserved

periphery are almost the same. The magnitude of difference between the local Nusselt numbers at the bottom and top at any given corresponding cross-section thus increases with increase in pitch. However, variation of local Nu for the coils with pitch of 45 and 60mm are identical. Nusselt number on the outer side of the coil is found to be the higher among all other points at a specified cross-section, while that at the inner side of the coil is the lowest. However, the average Nusselt number is not affected by the coil pitch.

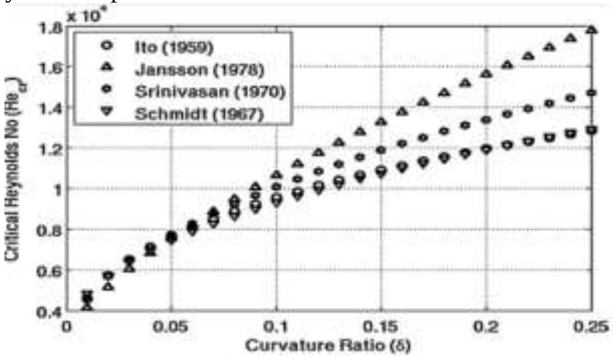

Figure 2.1 Critical Reynolds number predicted by various correlations [3]

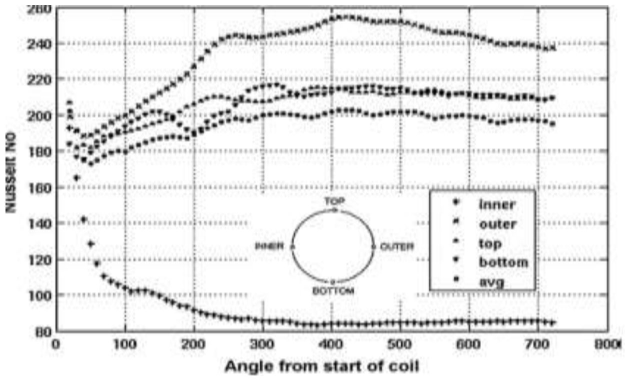

Figure 2.2 Variation of Nu along the length coil [3]

Shajikumar [5] presented an investigation on tube temperature distribution in a water tube boiler, performs detailed efficiency testing and simulation of thermal flow inside an industrial boiler. The simulations were carried out using commercial available CFD software. Figure 2.3 shows that temperature varies along the radial distance.

The analysis of the temperature distribution for every location inside the domain is conducted by setting constant heat fluxes, and varying parameters such as mass flow rate of steam, steam inlet temperature and scale thickness.

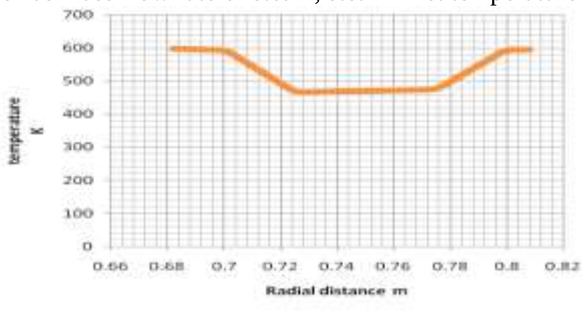

Figure 2.3 Temperature vs axial distance [5]

The temperature distribution in the boiler tube is affected by many variables such as mass flow rate of steam, steam temperature, feed water temperature and pressure. It is found that the increase of mass flow rate of steam through the boiler tube causes the decrease in temperature in the inner tube wall. This behavior occurs due to heat releasing from flue gas to steam is not proportional as the ability to absorb heat from flue gas for higher mass flow rate is faster. If mass flow rate of steam is increased, as a consequence of it, temperature of flue gas must be increased to make heat balance in equilibrium condition. The steam inlet temperature affects the thermal efficiency of a thermal power plant. The higher steam inlet temperature increases thermal efficiency. Higher operating temperature can also increase scale growth.

Begum[6] reported that now day’s boiler tube failures is main cause occurs in plant and that’s affects on total performance of the plant. The thermal heat transfer analysis was conducted by applying surface heat flux on the inner wall surface of the tube that will either supply heat or take away heat during the process. The vector option was then activated and the vector plot of the heat flow was displayed. The thermal fluctuation that occurred due to the slag and the change of operating conditions also creates alternating stresses. As a result the boiler tubes were subjected crack. The

© 2015, IERJ All Rights Reserved

plant need to be overhauled for maintenance purpose due to the leak generated from the cracks. The analysis is pre-dominantly simulation based which focused mainly on the root cause of the failure of boiler tubes exposed to operating

condition. The thermal heat transfer analysis was conducted by applying surface heat flux on the inner wall surface of the tube that will either supply heat or take away heat during the process. There were several failure modes that may occur in a boiler tube, i.e. stress-corrosion cracking, pitting, water-side corrosion, corrosion, fatigue failure, overheating, dissimilar metal weld fatigue, mechanical fatigue and erosion. In this research the failure modes of boiler and its end cracks due to the dissimilar metal weld is analyzed. Hence data were collected and explored to determine the cause of failure and as a solution is suggested. The use of suggestion in boiler tube can eliminate the crack occurrence or delay the process due to thermal properties. The prevention of crack will reduce frequent maintenance and thus the cost of operation will reduce.

Bingzhi LI [7] presented the deposition formed on super-heater tubes decrease the heat transfer. Deposits may also cause corrosion of tube material. In Kraft recovery boiler furnaces, the alkali-rich deposits are molten, and flow down along the super-heater tubes. In this work, a sub model which calculates the structure of a slag flow was implemented into a CFD model for super-heater tube sections. Specifically, it is applied to two tube bends on one of the first group of tube banks exposed to the flue gas. By taking into account deposition by inertial impaction, assuming all depositing particles fully molten and the steam temperature in super-heater constant, the slag model can provide thickness and surface temperature distributions of the owing deposit layer.

Khanorkar [10] presented CFD analysis of the vertical tube.Vertical copper tube having constant cross section area used for representing the medium through which natural convection of water takes place. In this work the study and analysis of natural convection flow of water through vertical pipe is done and study the effect of the physical parameters of tube like diameter, length and heat flux on the outlet flow parameters like velocity and temperature. Constant heat flux is boundary conditions provided on the entire tube surface. In this study he found that outlet temperature and outlet velocity that going to be increased as tube length was increased but as diameter of pipe increased outlet temperature also increased but velocity decreased as shown in the figure 2.4.

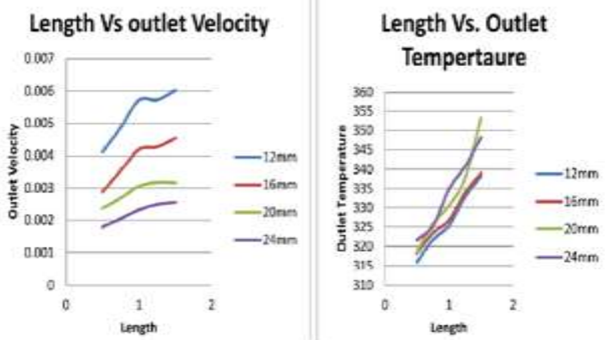

Figure 2.4 Results between length vs. outlet velocity and length vs. outlet temperature [10]

III. EXPERIMENTATION

3.1. Actual working parameter

Working Fluid	Steam
Inlet Mass flow Rate(single)	0.4320 Kg/s
Inlet Temperature	573 K
Outlet Pressure	40 Kg/cm^2
Constant Wall Temperature	873 K
Specific heat of steam(C_p)	2916.19 J/Kg-K
Thermal conductivity of steam(K)	0.05194 W/Kg-k
Density of Steam(ρ)	18.46 Kg/m3
Dynamic Viscosity of Steam (μ)	1.985e-5
Total length of Super heater pipe	28.07 m
Inner diameter of super heater pipe	0.041 m
Number of Super heater Pipe from the steam drum	45
Total surface heat flux	89200 W/m^2
Total heating Surface Area of Super heater	198 m^2

© 2015, IERJ All Rights Reserved

3.2 Computational Details

To find out the turbulent flow in the super heater pipe of a boiler, K-ϵ model is employed in this study. In the present work the finite volume modeling was conducted by the simulation Fluent 14.0. Velocity and Pressure field are linked by semi implicit method for pressure linked equation SIMPLE algorithm.

3.2.1 Continuity equation

$$\frac{D\rho}{Dt}+\rho\nabla.V=0 \quad (2)$$

3.2.2 Momentum Equations

X Component

$$\rho\frac{Du}{Dt}=-\frac{\partial p}{\partial x}+\frac{\partial\tau_{xx}}{\partial x}+\frac{\partial\tau_{yx}}{\partial y}+\frac{\partial\tau_{zx}}{\partial z}+\rho f_x \quad (3)$$

Y Component

$$\rho\frac{Dv}{Dt}=-\frac{\partial p}{\partial y}+\frac{\partial\tau_{xy}}{\partial x}+\frac{\partial\tau_{yy}}{\partial y}+\frac{\partial\tau_{zy}}{\partial z}+\rho f_y \quad (4)$$

Z Component

$$\rho\frac{Dw}{Dt}=-\frac{\partial p}{\partial z}+\frac{\partial\tau_{xz}}{\partial x}+\frac{\partial\tau_{yz}}{\partial y}+\frac{\partial\tau_{zz}}{\partial z}+\rho f_z \quad (5)$$

3.2.3. Energy Equation

$$\rho\frac{D}{Dt}\left(e+\frac{V^2}{2}\right)=\rho\dot{q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)-$$

$$\frac{\partial}{\partial x}(up)-\frac{\partial}{\partial y}(vp)-\frac{\partial}{\partial z}(wp)+\frac{\partial(u\tau_{xx})}{\partial x}+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(u\tau_{zx})}{\partial z}+$$

$$\frac{\partial(v\tau_{xy})}{\partial x}+\frac{\partial(v\tau_{yy})}{\partial y}+\frac{\partial(v\tau_{zy})}{\partial z}+\frac{\partial(w\tau_{xz})}{\partial x}+\frac{\partial(w\tau_{yz})}{\partial y}+\frac{\partial(w\tau_{zz})}{\partial z}+$$

$$\rho f.V \quad (6)$$

3.3 Assumptions in the study

The numerical simulations in this study have been performed based on some assumptions which are the assumptions of other researchers while investigate the super heater flow analysis. Following are some of the assumptions on which the current work is based.

- The fluid is assumed to be incompressible with constant thermo-physical properties and the flow is assumed to be 3 dimensional, turbulent, steady and non-rotating. Because of the working fluid is steam.
- Constant temperature is prescribed on the super heater tube wall.
- No-slip velocity conditions are applied at walls.
- A uniform mass flow rate and temperature are set at the inlet.
- Pressure outlet condition is assumed at the outlet.
- A turbulence intensity level of 1.00% is assumed for the flow.

3.4. Experimental to numerical Conversion

Fig 3.3 shows how the experimental geometry converted to numerical model .In case of the numerical model only the flow has been simulated. The constructions of outer portions, walls etc. as in case of experimental set up have not been drawn as it is not required in numerical study, therefore here simulate flow inside the super heater.

Fig.3.1 Experimental Model

© 2015, IERJ All Rights Reserved

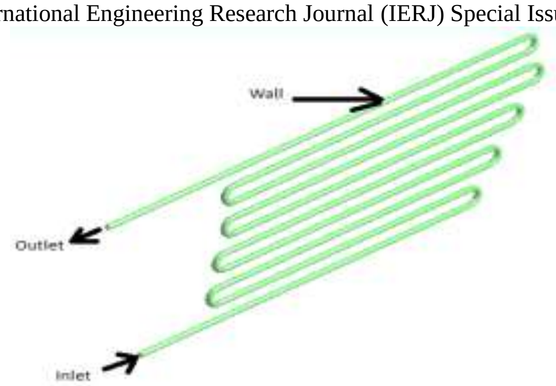

Fig.3.2 Numerical Model in this study

Here experimental model is converted in to numerical model by utilizing flow governing equations.

3.5 Boundary conditions

The numerical simulations in this study have been performed based on some assumptions of other researcher while study the super heater flow analysis. Following are some of the assumptions on which the current work is based.

a) The fluid is assumed to be incompressible with constant thermo-physical properties and the flow is assumed to be 3 dimensional, turbulent, steady and non-rotating. The working fluid is steam.

b) Constant temperature is prescribed on the wall of super heater tube.

c) No-slip velocity conditions are applied at all walls.

d) A uniform mass flow rate and temperature are set at the inlet

e) A pressure outlet condition is assume at the outlet.

f) A turbulence intensity level of 1% is assumed for the flow.

3.6 Mesh Conversions Study

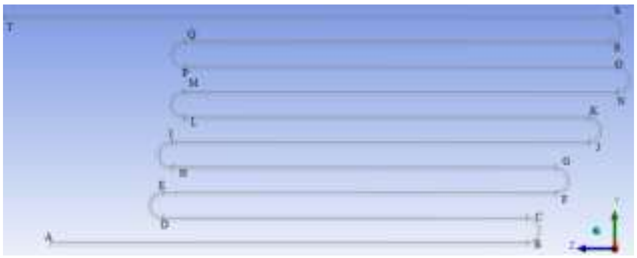

Fig:3.3 Numerical Model of Super Heater for conversions study

Here we divide geometry in nighteen parts to study the various thermal parameters in the different parts of the super heater pipe as shown in the fig, the fluid flow from inlet to the outlet. Here we drawn the line inside the super heater geometry at the center and measure the temperature and pressure at the various parts in the super heater pipe , measure the pressure, temperature and drawn the curves for the section ST .

3.6.1 Various plots for all grids

To ensure the accuracy and validity of the numerical result, a careful check of the grid dependence of the numerical solutions has been carried out by considering 4 grid systems with large number of grid points, i.e., 64932cells, 63900 cells, 50048 cells and 42080 cells for the simulations. The pressure, temperature at the center line of the super heater tube surface nusselt number and skin friction coefficient for the wall of the super-heater from these 4 grid systems is plotted.

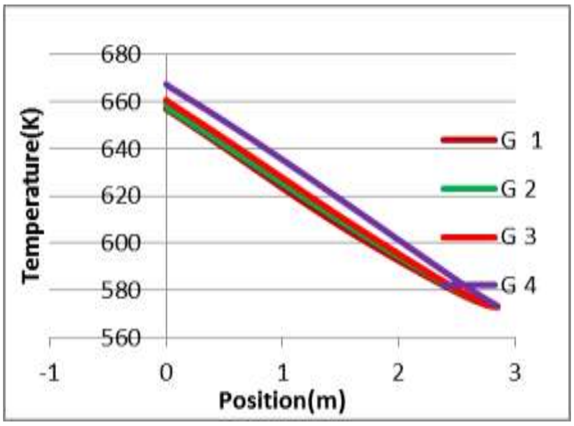

Fig.3.4 Temperature Plots for all grids of Super heater Section AB.

© 2015, IERJ All Rights Reserved

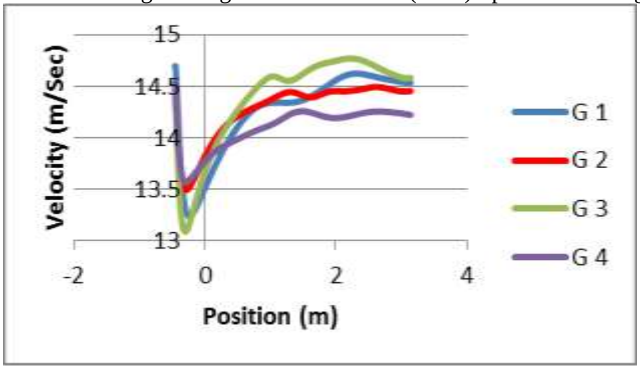

Fig.3.5 Velocity Plots for all grids of Super heater Section ST

Figure 3.5 shows that the velocity is changed at the bending section of the super heater, if a fluid is moving with a straight pipe that after few point becomes curve, the bend will cause the fluid particles to change their main direction of motion.

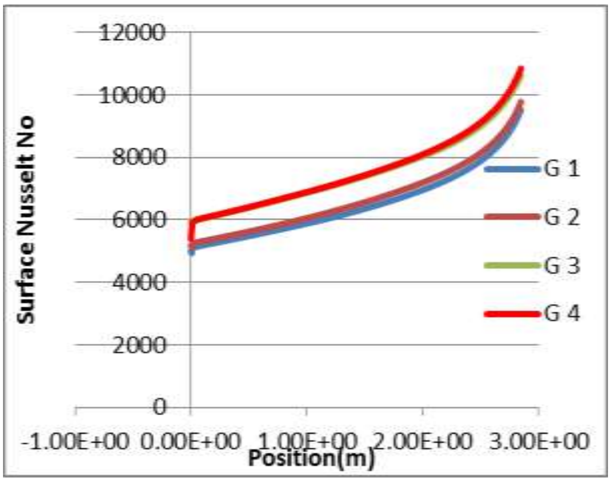

Fig.3.6 Surface Nusselt number for all grids of a super heater Section AB

The surface nusselt number is decrease along the length of the super-heater and grid 3 and grid 4 show the same result shown in the fig 3.6.

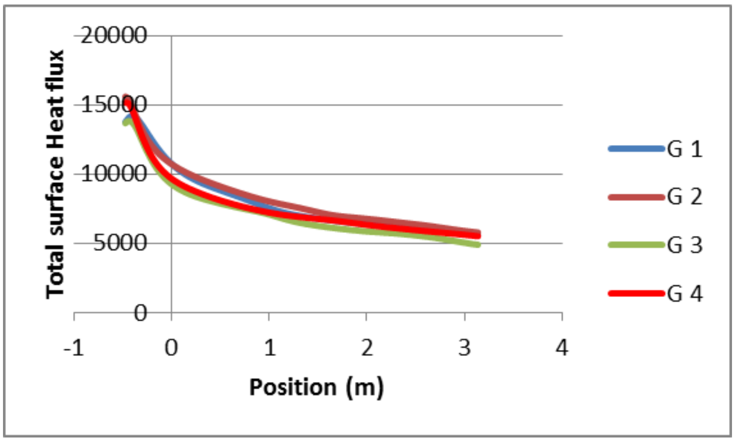

Fig 3.7 Total surface heat flux for all grids of a super heater Section ST

Total surface heat flux is decrease along the length of super-heater and its magnitude is minimum at section ST of super heater wall. It is found that the normal deviation of temperature and pressure, surface nusselt number, total surface heat flux and skin friction coefficient from 50048 cells to 63900 cells, 42080(i.e. Grid 2, Grid 3 and Grid 4) .Thus, to save computer resource and keep a balance between computational economy and prediction accuracy, the grid with 50048 cells is chosen for the simulations. These simulations have performed on a computer having a frequency of 2.3 GHz and a core memory of 2 GB.

3.7 Convergence History

The convergence criterion kept for continuity, momentum, k and ϵ equations and energy equation are 10^{-6}. The Solution is converged at 164 iterations and got following results. 5000 iterations are given at start of the solution and solution converges at 164 numbers iterations. Parameters like temperature, velocity at outlet and surface nusselt number and skin friction coefficient monitors are set in fluent monitor setup.

© 2015, IERJ All Rights Reserved

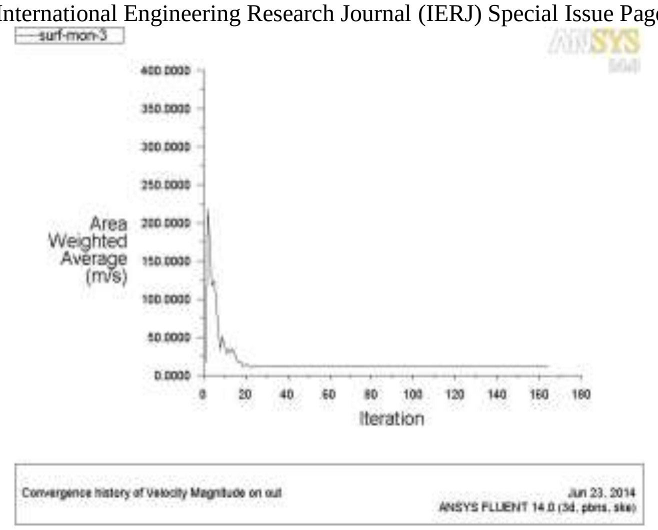

Fig. 3.8 Conversions history of velocity at outlet of super heater

The velocity fluctuations also takes place at the outlet of the super heater it increases up to 10 iterations then decreases and after 20 iterations it remains constant up to 164 iterations, as shown in the figure 3.8.

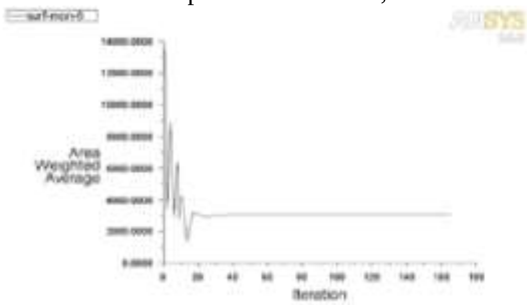

Fig. 3.9 Convergence history of Nusselt number along the wall of super heater

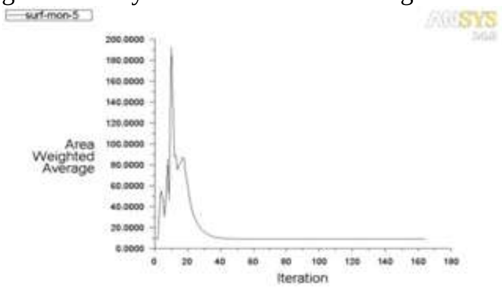

Fig. 3.10 Conversions History of skin friction coefficient along the wall of super heater

Figure 3.10 shows that surface nusselt number along the wall of super heater first increases then decreases up to 20 iterations after 20 iterations it remains constant up to 164 iterations. Skin friction coefficient along the wall of the super heater first increase up to 20 iterations then decrease and from iteration 40 remains constant up to 164 iterations as shown in the fig. 3.10

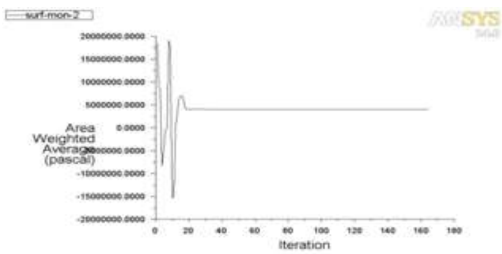

Fig.3.11 Conversion history of pressure inlet of super heater pipe

From the figure 3.11 it is observed that pressure at inlet of super heater is fluctuate from iteration 1 to iteration 20 after 20 iterations it remains constant

IV. RESULTS AND DISCUSSION

4.1. Various curves for super heater

Here various curves are plotted on the line drawn inside the geometry of the super heater to study thermal flow inside super heater.

© 2015, IERJ All Rights Reserved

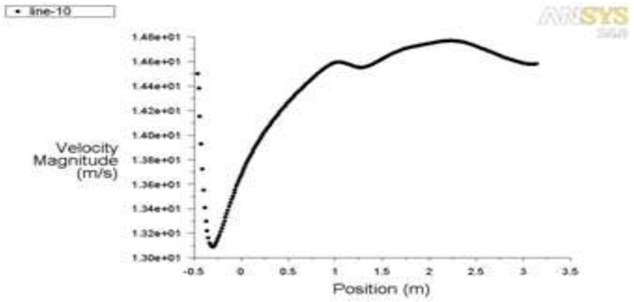

Fig. 4.1 Velocity plot for Super heater Section ST

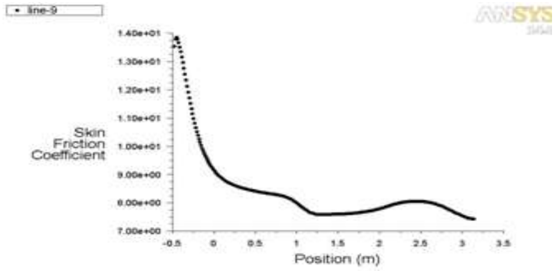

Fig. 4.2 Wall skin friction coefficient plot for Super heater section ST

From the figure 4.2 it is observed that the velocity is increased along the length of the super heater then slightly decreased. Velocity is high at bending portion of super heater pipe. There will be an adverse pressure gradient generated from the curvature with an increase in pressure, therefore a decrease in velocity close to the convex wall, and the contrary will occur towards the outer side of the pipe. Skin friction coefficient also high at the bending section of the super heater and decreases along the wall of super heater as shown in figure 4.3.

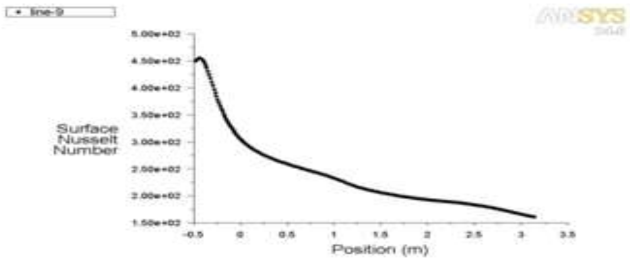

Fig. 4.3 Surface Nusselt number plot for Super heater section ST

The surface nusselt number is decreases from inlet to outlet of super heater wall. It is minimum at the super heater of a wall at outlet as shown in the figure 4.4. & Figure 4.5 show that temperature is increased with the super-heater section ST. Temperature along super heater section ST temperature increases from 865°K to 868°K. Highest temperature occurs at the outlet of super heater that is 868 °K.

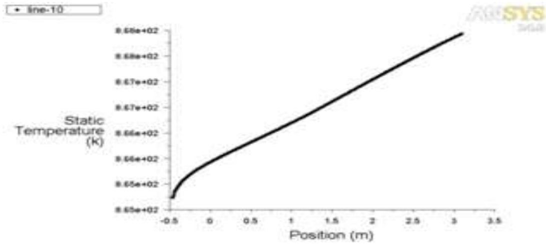

Fig. 4.4 Temperature plot for super heater section ST

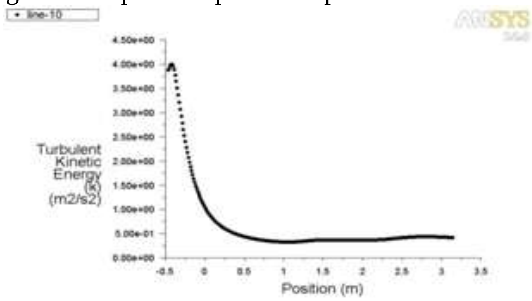

Fig. 4.5 Turbulent kinetic energy plot for super heater section ST

© 2015, IERJ All Rights Reserved

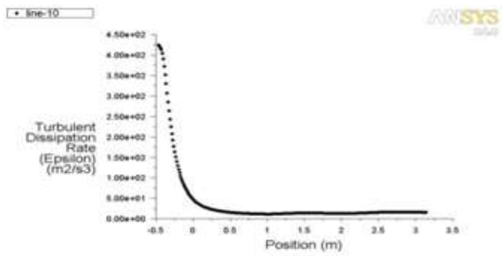

Fig. 4.6 Turbulent dissipation rate plot for super heater section ST

Figure 4.5, figure 4.6 shows that turbulent kinetic Energy, turbulent dissipation rate decreases along the length of super heater section ST. Turbulent kinetic energy and turbulent dissipation rate is maximum at the bending section of super heater.

V. VARIOUS CONTOURS FOR SUPER HEATER

Here various contours are plotted for a super heater to study and understand the thermal flow in the pipe to resolve the operational problems. From figure it is observed that temperature near the U-tube bend found more than another part of the super heater. Temperature in the super heater section ST, SR, QR and PQ shows the same temperature. Temperature is decreases along the length of super heater. This fluctuation of temperature may cause the boiler tube leakage. As the super-heater is the heat exchanger it increases the temperature of the steam flowing inside the tube and so the temperature of steam increases from inlet to outlet due to the super heater wall is heated by flue gas around 600 ° Celsius. At certain region sudden temp variation leads to the thermal embrittlement. This may leads to fatigue and cracks near the joint of the tube resulting into change in shape and cracks near the corners.

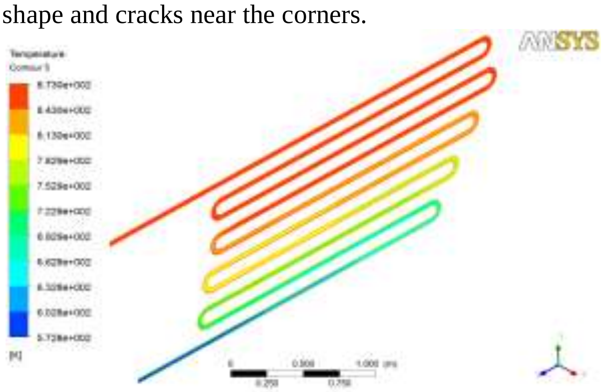

Fig.5.1 contour of temperature along the interior of super heater

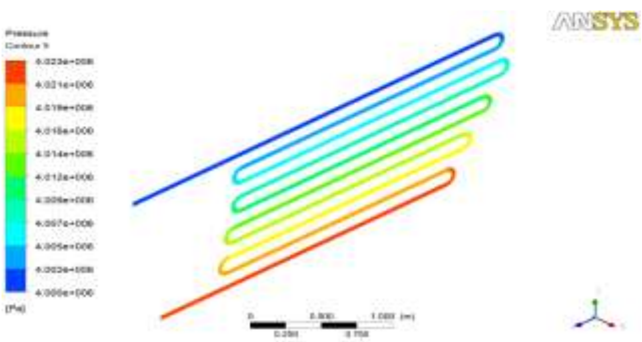

Fig.5.2 contour of pressure along the interior of the super heater

Fig. 5.1 show that pressure is decreases along the length of super heater, maximum pressure occurs at inlet and minimum pressure at outlet. The pressure drop in various bent geometries. The pressure drop is more significant due to flow separation at the inner wall in elbows as compared to bends.

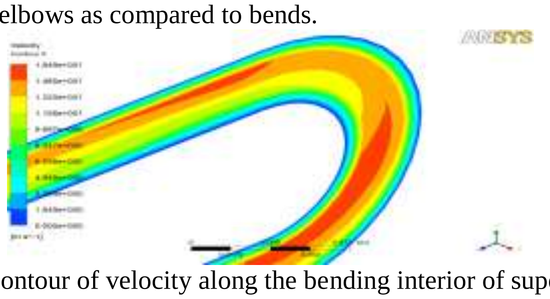

Fig.5.3 Contour of velocity along the bending interior of super heater

© 2015, IERJ All Rights Reserved

From the figure 5.4 and figure 5.5 it is observed that the velocity fluctuation takes place at the U-bending section. The sudden increment and decrement of the velocity takes place at the bending part of the super heater. Maximum velocity observed at the bending of a super heater. A boundary layer is formed as the fluid enters the pipe where the viscous forces are confined while the core is inviscid, like in a straight pipe. The secondary flow generated by the curvature is therefore moving the slower fluid from the boundary layer inwards and the faster fluid at the outwards. The inflow condition greatly affects the initial development of the flow with non-uniformity in wall shear stress, i.e. the shear is largest at the inner wall before the maximum moves to the outer wall, appearing at two times larger distance for the first inlet condition than for the second one.

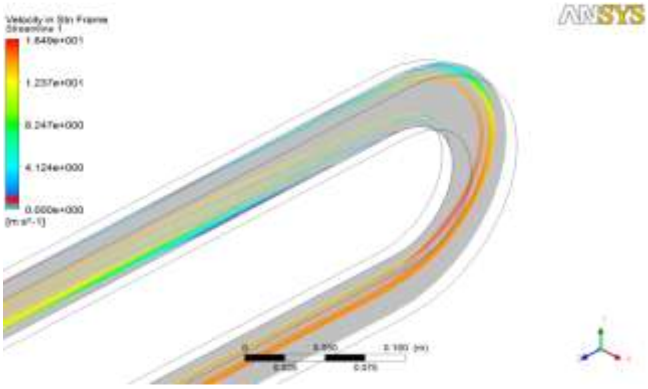

Fig.5.4. Contour of velocity streamline along the bending interior of super heater

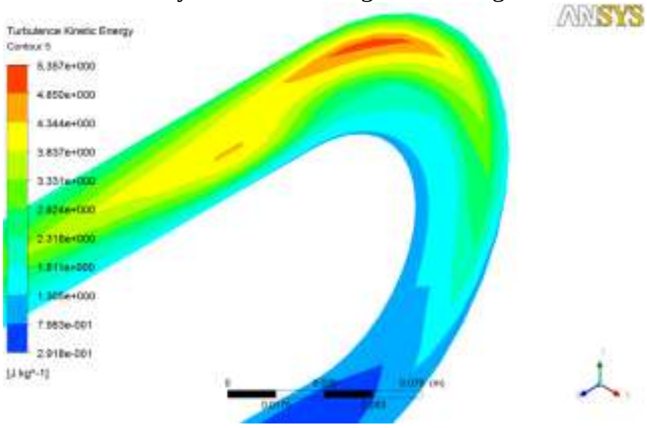

Fig.5.5 Contour of turbulent kinetic energy along bending interior of super heater

VI. VARIOUS PLOTS OF SUPER HEATER FOR PARAMETRIC STUDY

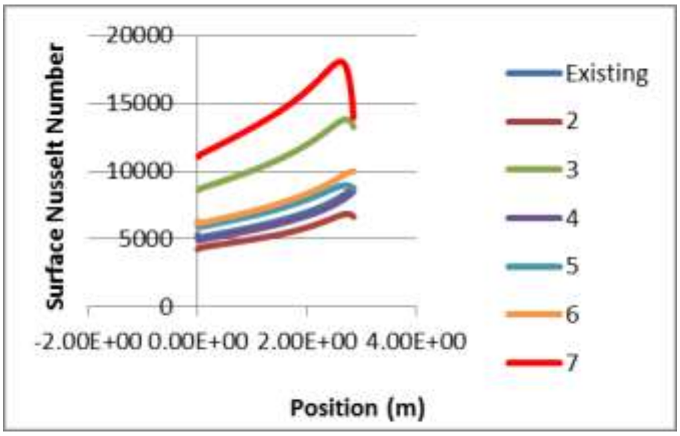

Fig.6.1 Nusselt number comparisons along the wall of super heater section AB

From the figures 6.1 is found that if the mass flow rate of steam increases then nusselt number increases. If the total number of tubes of super heater decreases then mass flow rate and nusselt number increase. If the diameter of super heater tube increases then nusselt number also increases.

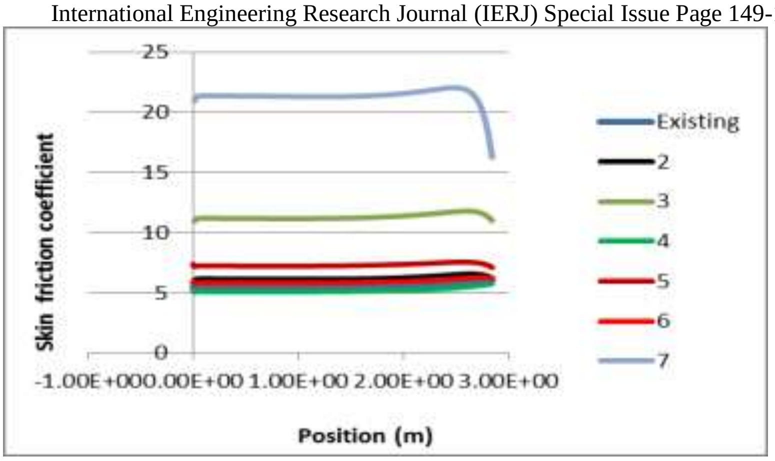

Fig.6.2 Skin friction coefficient comparisons along the wall of super heater section AB

If the diameter of super heater decreases then skin friction coefficient along the wall of super heater increase due to this pressure drop also increased at the bending section of super heater. If the mass flow rate of steam decreases then skin friction coefficient along the wall of super heater decrease as shown in the figure 6.2. The diameter of super heater tube increases, the heat transfer also increase.

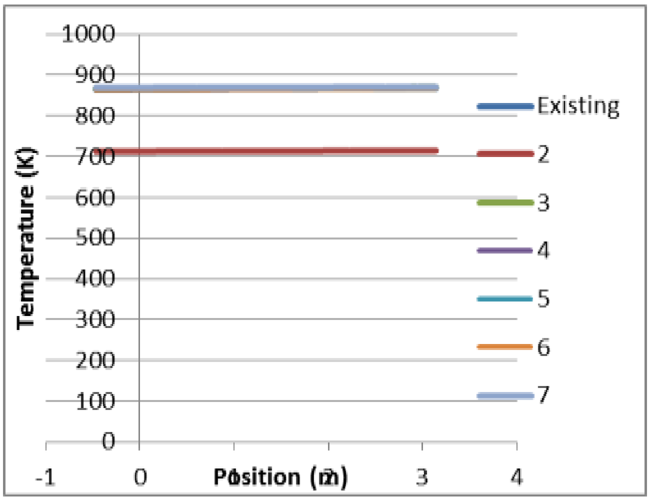

Fig.6.3 Temperature comparisons of a super heater at ST

It is evident that the mass flow rate strongly affects the temperature distribution of the water tube boiler. From figure 6.3 is found that the increase of mass flow rate of steam through the boiler tube causes the decrease in temperature in the inner tube wall. This behavior occurs because of the heat release from flue gas to steam is not proportional as the ability to absorb heat from flue gas for higher mass flow rate is faster. If mass flow rate of steam is increased consequence of it, temperature of flue gas must be increased to make heat balance in equilibrium condition. The higher steam inlet temp increases thermal efficiency but operating boiler with higher temperature has some disadvantages (i.e. to make steam inlet temp higher, more time is required, and also strength of tube materials should be considered as higher temperature will degrade the strength of material and thermal conductivity).

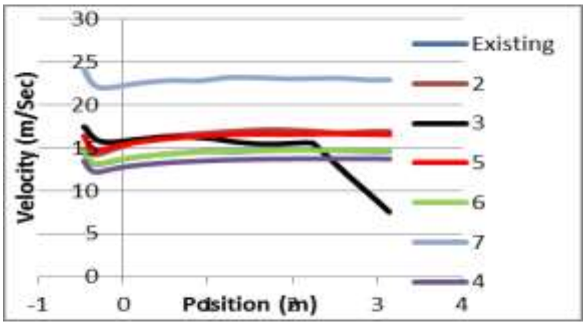

Fig.6.4 Velocity comparisons of a super heater section ST

© 2015, IERJ All Rights Reserved

Velocity of steam is depends on diameter of the super heater pipe. From figure 6.4 it cleared that if diameter the super heater tube increased then velocity of a steam decreases. Also cleared that velocity of steam is maximum at lower diameter of super heater pipe.

VII. CONCLUSION

From above results and study concluded that
1. For existing super heater the heat transfer decreases along the length of a super heater pipe, and temperature range, surface Nusselts number is also same at sections OP ,QR and ST of a super heater pipe. Thus the super heater length can be reduced to avoid thermal losses as well as financial losses.
2. If the mass flow rate of steam in super heater is decreased then temp of the steam inside super-heater tube increases.
3. If the total number of super heater pipes is decreased by 5 then mass flow rate increases in the tube so that average Nusselt number increases. The heat transfer increases and temperature decreases which assists to save the super heater from overheating. Turbulent kinetic energy at bending interior is increases due to pressure drop increases
4. If the mass flow rate is increased Pressure drop increased. Average Nusselt number decreases and turbulent kinetic energy at bending interior is increases.

REFERENCES

[1] Raja Saripalli, May 2005, “Simulation Of Combustion And Thermal Flow in an Industrial Boiler” , ESL-IE-05-05-42 , Proceedings of the Twenty-Seventh Industrial Energy Technology Conference, New Orleans, LA.
[2] Rahimi Masoud, Abbas Khoshhal and Syed Mehdi Shariati, (2006), “CFD modeling of boiler tubes rupture ” Journal of Applied Thermal Engineering, 26, 2192-2200.
[3] J.S. Jayakumar, February 2009, “CFD analysis of single-phase flows inside helically coiled tubes” Elsevier Ltd., Computers and Chemical Engineering 34 (2010) 430–446
[4] Ajay N. Ingale, March 2012 , “CFD Analysis of Super heater in View of Boiler Tube Leakage”, International Journal of Engineering and Innovative Technology (IJEIT) Volume 1,ISSN: 2277-3754.
[5] Shaji kumar K.R , Sep-Oct. 2012, “An Investigation on Tube Temperature Distribution in a Water Tube Boiler” IOSR Journal of Mechanical and Civil Engineering (IOSRJMCE) ISSN: 2278-1684 Volume 2, pp 45-50.
[6] Shahida Begum, 2012, “Analysis of End Crack in Boiler Tube’’ Advanced Materials Research Vol. 576 , pp 749-752.
[7] Bingzhi Li, January 2013, “CFD investigation of slagging on a super-heater tube in a kraft recovery boiler” Fuel Processing Technology Volume 105, pp 149–153.
[8] Raquel Antunes Hess, 2013, “Analysis of Heat Transfer and overheating failures associated with wall temperatures in Steam Superheaters”, 22nd International Congress of Mechanical Engineering, pp 3-7.
[9] Prashant M Khanorkar, 2013, “Cfd Analysis of Natural Convection Flow Through Vertical Pipe” Int. J. Mech. Eng. & ISSN 2278 – 0149, vol. 2, No. 3.
[10] Raja Saripally,Ting Wang and Benjamin Day, 2005, “Simulation of combustion and thermal flow in an industrial boiler”, Proceedings of 27th Industrial Energy Technology Conference, New Orleans, Louisiana .

© 2015, IERJ All Rights Reserved