CFD ANALYSIS OF WASTE HEAT BOILER

SREEKANTH S[1], Dr. BENNY PAUL[2]

***Abstract*—Waste Heat boilers are used to recover waste heat from high temperature exhausts in chimney stacks. Waste heat boilers are typically water tube boilers which use large volume, high temperature waste heat streams as a heat source as opposed to conventional fuel. Typical heat sources include hot exhaust gases from such equipment as gas turbines, incinerators, furnaces and reciprocating engines. Should the waste heat in exhaust gases be insufficient for generating the required amount of process steam, it is sometimes possible to add the auxiliary burners. These systems burn fuel in the waste heat boiler or an afterburner may be added to the exhaust gas duct just ahead of the boiler.In this analysis, flow simulation (CFD) of the Regenerative Air (RA) & Gas Turbine (GT) ducts along with inlet pipes and mixing chamber were carried out for various configurations to identify the real cause for the vibrations developed. An amicable solution identified was to guide the flow properly through the RA duct with guide plates to eliminate the vortex generation and circulation induced which was the primary cause for vibrations developed in the duct assembly.**

***Index Terms*—CFD (Computational Fluid Dynamics), Waste Heat Boiler, RA (Regenerative Air), GT(Gas Turbine).**

I. Introduction

Waste heat boilers are ordinarily water tube boilers in which the hot exhaust gases from gas turbines, incinerators, etc., pass over a number of parallel tubes containing water. The water is vaporized in the tubes and collected in a steam drum from which it is drawn out for use as heating or processing steam.

Because the exhaust gases are usually in the medium temperature range and in order to conserve space, a more compact boiler can be produced if the water tubes are finned in order to increase the effective heat transfer area on the gas side The flue gas temperature, pressure and velocity field of fluid flow within an economizer tube using the actual boundary conditions have been analyzed using CFD tool.

Manuscript received September, 2015.

***Sreekanth S**, PG Scholar, Department of Mechanical Engg, Mar Athanasius College of Engineering, Kothamangalam 9961822511*

***Dr Benny Paul**, Professor, Department of Mechanical Engg, Mar Athanasius College of Engineering, Kothamangalam*

II. LITREATURE REVIEW

Krunal P. Mudafale & Hemant S. Farkade[1] work on "CFD analysis of economizer in atengential fired boiler".A.D.Patil, P.R.Baviskar, M.J.Sable, S.B.Barve[2] work on "To optimize economizer design for better performance" This paper focuses on optimization of economiser design with finned & baretube economiser. The aim of this work is to develop methodology which finds optimisation of economiser design. TSUNG-FENG WU[4] work on "failure analysis for economizer tube of the waste heat". This paper is about failure analysis of the leakage of the economizer tube of the waste heat boiler inthe energy factory. The results show that although the material and mechanical properties of thefailed tube, were inferior to those of the new one, most of them were still satisfactory to the criterion requirement it is clear that the crack initiated in the outer surface and propagated toward the innersurface of the tube and the crack was identified to be rectangular in shape. Deendayal Yadav, Dr. G. V. Parishwad, P. R. Dhamangaonkar*, Dr. S. R. Kajale, Dr. M. R.Nandgaonkar, Dr. S. N. Sapali.[5] work on "effect of arreasters on erosion in economizer zone and its analysis". The authors in this paper have attempted to suggest a probable solution for reduction of erosion in economizer zone and its analysis using CFD tool. In this paper the authors have submitted the findings of analysis of finned tube economizer with Arresters at different inclinations distribution. Saripally et al. (2005) conducted a computational fluid dynamics simulation of thermal flow in an industrial boiler. Simulation has been done to understand the thermal flow inside the boiler tube to resolve the operational problem and search for optimal solution. During the analysis, the geometry of the system and boundary conditions such as inlet velocities and flow rate was changed to view their effect on thermal flow patterns or species concentration. In this analysis, flow simulation (CFD) of the Regenerative Air (RA) & Gas Turbine (GT) ducts along with inlet pipes and mixing chamber is carried out for various configurations to identify the real cause for the vibrations developed.

ISSN: 2278 – 7798

III. MODEL DESCRIPTION

This chapter introduces the reader to the setup used in the experiments. The important and unique aspects of the setup are described:

A.PROBLEM ANALYSIS

At the plant location, vibration problem for the transition duct between the gas turbine and the Waste Heat Boiler (WHB) was identified and a solution was sought. For this, various data needs to be collected so that the actual cause of vibration can be pointed out and a proper solution can be framed. The input data was collected from the plant and the problem statement was identified based on discussions for detailed investigation, solution and reporting.

Input study revealed variations in the operating parameters (mass flow rate) when Operational Trends was compared with PFD data. As per PFD data the mass flow rate need to be 337kg/s & 147kg/s in the RA &GT duct respectively. But the Operational Trends recorded a maximum mass flow rate of 1073kg/s & 62kg/s in the RA & GT duct respectively.

Based on this observation it was proposed to study the flow physics occurring the RA & GT ducts both as per PFD data & Operational Trends. This approach will help to identify the existing operational problems of the Waste Heat Boiler (WHB) Transition Duct.

FLUID	FLUE GAS
Temperature at RA & GT inlets	857 K & 901 K
Mass Flow Rate at RA & GT inlets	337 Kg/s & 147 Kg/s
Density (ρ)	0.6214.5 kg/m^3
Viscosity (μ)	0.0284 Kg/ m-s
Conductivity (k)	0.042749 W/m−K
Specific Heat (C_p)	1099.87 J/kg−K

Table.3.1Input data

B.GEOMETRY CREATION

After careful analysis of the problem, a geometry of the desired part needs to be created for further investigation. Based on the information from input data a geometry was constructed using the software Catia V5 R20.

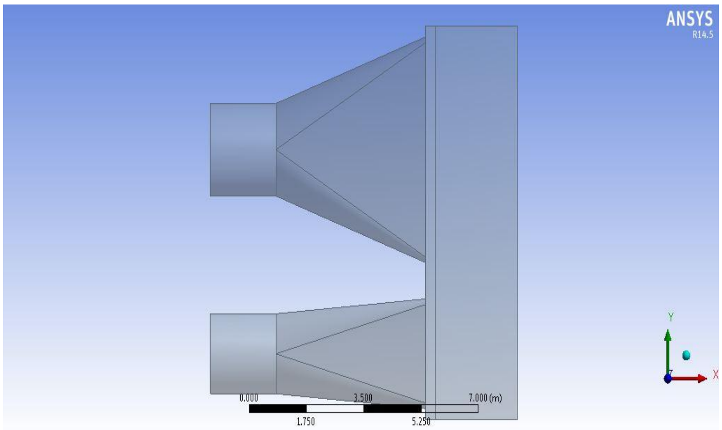


Fig.3.1Geometry created using Catia V5 R20

C.MESHING

The created geometry is meshed using Ansys ICEM CFD. The meshed geometries are shown below.

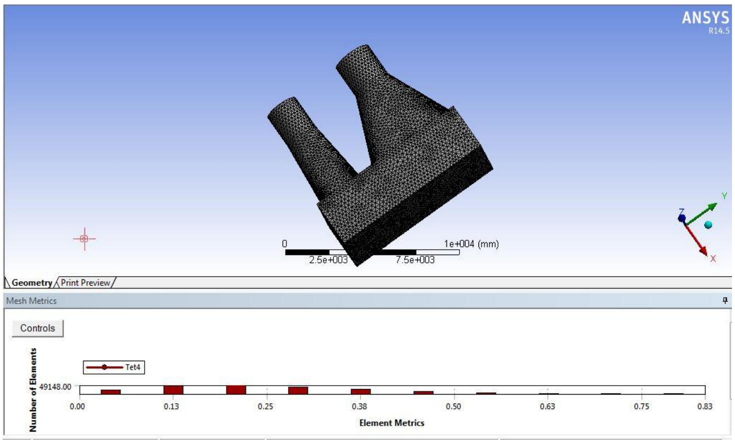


Fig.3.2The meshed geometry

The meshed geometry is used as the input for CFD software fluent for flow simulation.

D. ROOT CAUSE ANALYSIS AND MODIFIED GEOMETRY

Computational domain contains around 46001elements & 45122 nodes. Relevance centre is set as fine and smoothing is high and 60 is given because after that value there is no remarkable change in temperature. All other settings are to be done as default**.** After the cfd analysis the velocity contours and temperature contours of the flow through duct are obtained as follows:

ISSN: 2278 – 7798

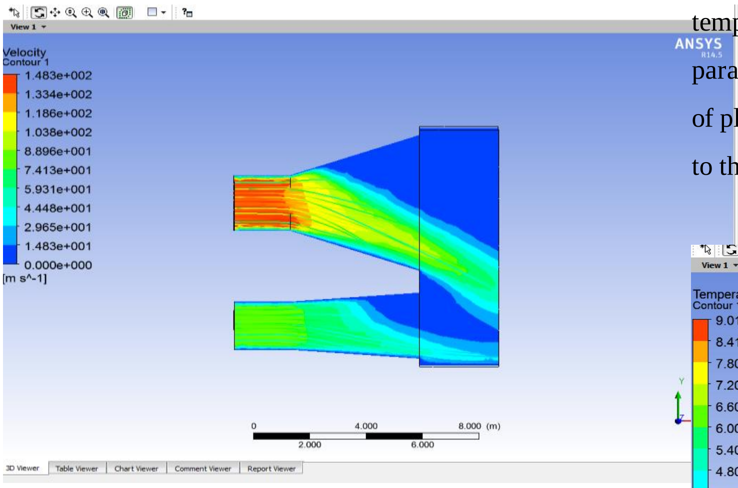


Fig.3.3Velocity contour

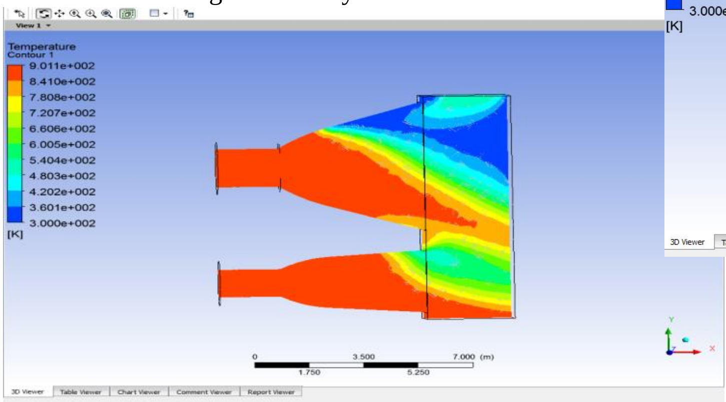
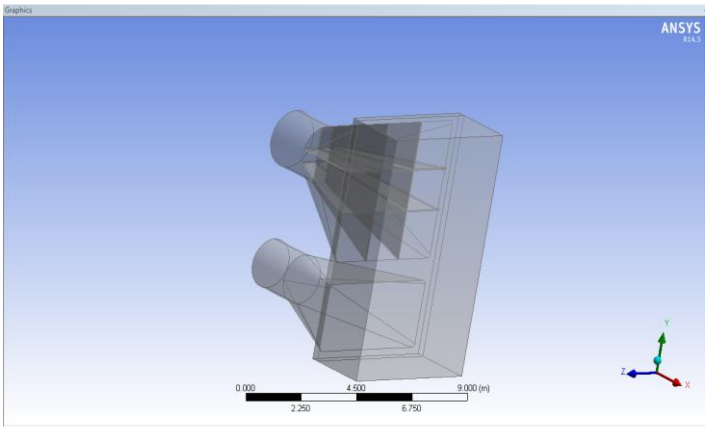


Fig3.4Temperature contour

From the above results it is clear that the flow is directed towards the bottom of the duct abnormally. To modify the flow into a uniform one, two sets of guide plates are introduced inside the duct.



Modified geometry with plates

E.FLOW ANALYSIS OF MODIFIED GEOMETRY

The modified flow is analyzed using ansys fluent with same set of solver settings and boundary conditions such as the mass flow rate, need to be 337kg/s & 147kg/s in the RA &GT duct respectively and the following flow simulation is obtained. From the temperature contour it is clear that the addition of plate have resulted in uniform distribution of gas temperature inside the duct. Since we have not changed any parameter used in the base geometry other than the addition of plate, we can credit this uniform temperature distribution to the modified geometry.

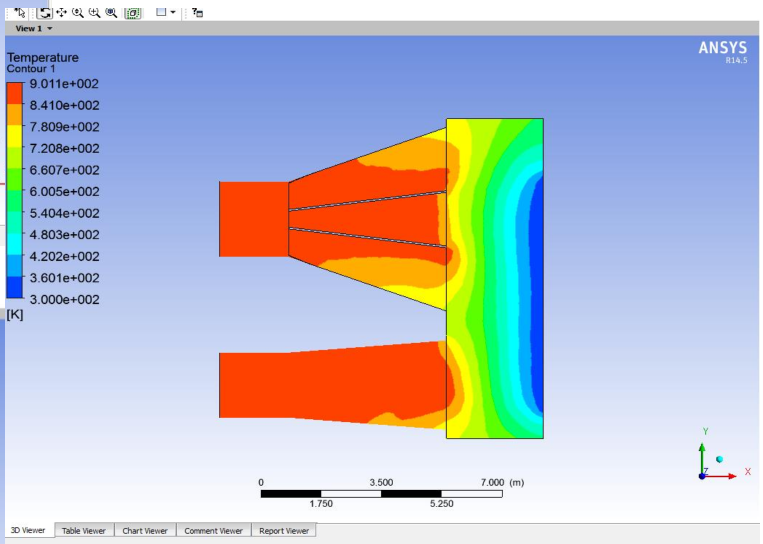


Fig.3.5Temperature contour

IV.RESULTS AND DISCUSSION

In this chapter we analyse the results obtained with the help of computational fluid dynamics (CFD). The thermal and fluid flow characteristics of the present problem were numerically simulated by computational fluid dynamics (CFD) technique. Commercial package ANSYS 14.5 is used for the present study. As per the data the mass flow rate need to be 337kg/s & 147kg/s in the RA &GT duct respectively. But the Operational Trends recorded a maximum mass flow rate of 1073kg/s & 62kg/s in the RA & GT duct respectively.

Based on this observation a study on the flow physics occurring inside the RA & GT ducts both as per data & Operational Trends were carried out. This approach helped to identify the existing operational problems of the Waste Heat Boiler (WHB) Transition Duct.

The existing structure inclusive of Inlet pipes, RA duct, GT duct & Mixing Chamber (MC) was subjected to analysis with and without Super Heater (SH))section. As a part of our study various configurations were analyzed for identifying the

ISSN: 2278 – 7798

causes for vibration and to suggest an optimized solution resulting in mitigation of induced vibration.

A suitable solution to the vibrations developed was to reduce the turbulent flow of gas inside the duct.For this a suitable solution has to be found. One of the most convenient ways of getting this done was the use of guide plates inside the duct. This will help us in reducing the cost as well vibrations without heavy modification of the duct.

First the CFD Analysis for an existing design of the RA & GT Duct to get the details of the Turbulence happening in the Ducts were done. Then Remodeling the existing model by adding plates once longitudinally and then transversally in the Divergent Suction to Check for occurrence of turbulence were done followed by its CFD Analysis. During the analysis the number of elements can be varied, when the number of elements increases , the accuracy of the results also increases .when the elements in the meshing are low which results in lowering the accuracy .

One of the important result obtained as a result of the remodeled design was evident in the velocity streamline as shown in the figure.Here we can see that the insertion of guide plates have given way for the turbublent flow to be transformed into streamline flow.As a result the vibrations got reduced and the life of RA duct, GT duct & Mixing Chamber (MC) were improved.

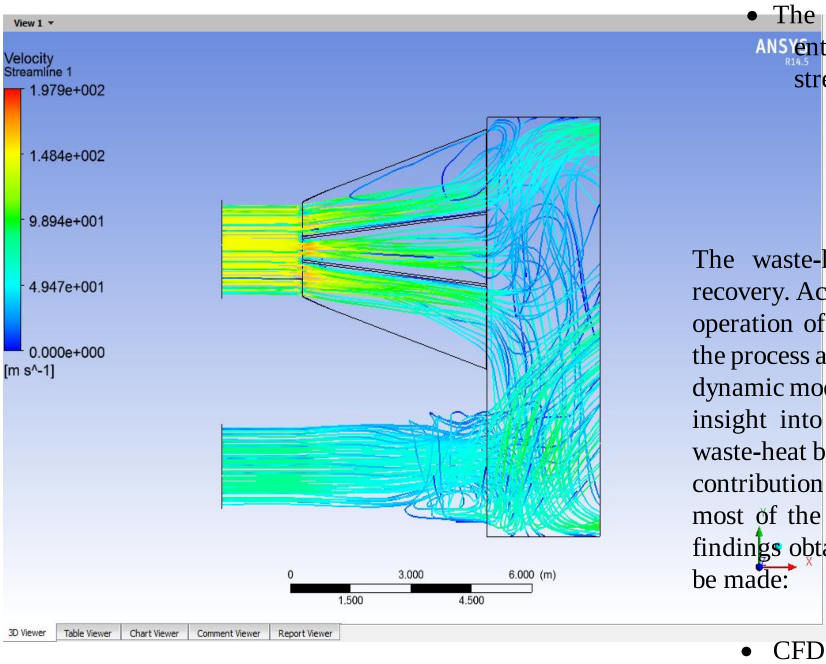


Fig.4.1Velocity streamline

V.SCOPE

a) Run the CFD Analysis for an existing design of the RA & GT Duct to get the details of the Turbulence happening in the Ducts.

b) Remodelling the existing model by adding plates once longitudinally and then transversally in the Divergent Suction to Check for occurrence of turbulence.

c) A better optimization of these plates have to be done in order to achieve the flow uniformity in the duct.

d) Remodelling with plates both longitudinal and transverse and to run the solver.

e) Conforming whether the design is satisfying the required criteria.

VII .VALIDATION OF PFD DATA

CFD analysis for verification of the mass balance of the RA duct-GT duct-MC section was carried out to validate the PFD data as a part of input study.

- The purpose of this analysis was to carry out the material and mass balance across the RA duct, GT duct and the Mixing chamber.
- The Feed Streams conditions were chosen so as to match the PFD and is one among the various analysis results obtained from the client end
- The objective was to check mass balance of the stream 3 from the given values of both stream 1 and stream 2.
- The stream 1 & 2 properties from PFD data were entered onto a virtual simulating software and stream 3 parameter were calculated as output.

VII. CONCLUSION

The waste-heat boiler is the common choice for heat recovery. According to the literature it is clear that successful operation of the boiler requires sufficient understanding of the process and its operating conditions. Computational fluid dynamic modeling did prove to be a valuable aid in providing insight into the flow and temperature profiles within the waste-heat boiler. It was possible to identify and quantify the contribution of turbulence and radiation effects. However, most of the analysis was done qualitatively. Based on the findings obtained, the following significant conclusions can be made:

- CFD Analysis for an existing design of the RA & GT Duct were carried out to get the details of the Turbulence happening in the Ducts.

ISSN: 2278 – 7798

- CFD Analysis of the remodeled design with guide plates were carried out to check for occurrence of turbulence.

- The placement of guide plates did have the desired effect of further enhancing the heat transfer while also distributing the off-gas flow more evenly through the waste-heat boiler duct.

VIII.REFERENCES

[1] Krunal P. Mudafale & Hemant S. Farkade CFD analysis of economizer in a tengential fired boiler, International Journal of Mechanical and Industrial Engineering (IJMIE) ISSN No. 2231 –6477, Vol-2, Iss-4, 2012.
[2] A.D.Patil, P.R.Baviskar, M.J.Sable, S.B.Barve to optimize economizer design for better performance, New aspects of fluid mechanics, heat transfer and environment isbn: 978-960-474-215-8.
[3] A.D.Patil, P.R.Baviskar, M.J.Sable, S.B.Barve Optimization of Economiser Design for the Enhancement of Heat Transfer Coefficient, International Journal of Applied Research In Mechanical Engineering (IJARME), ISSN: 2231 –5950 Volume-1, Issue-2, 2011.
[4] TSUNG-FENG WU Failure Analysis for the Economizer Tube of the Waste Heat Boiler.
[5] Deendayal Yadav, Dr. G. V. Parishwad, P. R. Dhamangaonkar*, Dr. S. R. Kajale, Dr. M. R.Nandgaonkar, Dr. S. N. Sapali Effect of Arresters on Erosion in Economizer Zone and itsAnalysis, AMAE Int. J. on Production and Industrial Engineering, Vol. 01, No. 01, Dec 2010.
[6] A.N. Aziz, P. Siregar, Y.Y. Nazaruddin, and Y. Bindar Improving the Performance of Temperature Model of Economizer Using Bond Graph and Genetic Algorithm, International Journal of Engineering & Technology IJET-IJENS Vol: 12 No: 01.
[7] N.S. Venkatesh Kumar and Prof. K. Hema Chandra Reddy, "CFD Analysis of Wind Driven Natural Cross Ventilation for a Generic Isolated Building", International Journal of Mechanical Engineering & Technology (IJMET), Volume 4, Issue 5, 2013, pp. 200 - 207, ISSN Print: 0976 – 6340, ISSN Online: 0976 – 6359.
[8] Sanjay Kumar Patel and Dr. A.C. Tiwari, "Performance Analysis of Supercritical Boiler", International Journal of Mechanical Engineering & Technology (IJMET), Volume 3, Issue 2, 2012, pp. 422 - 430, ISSN Print: 0976 – 6340, ISSN Online: 0976 – 6359.

Sreekanth S, PG student, Mechanical(Thermal Power Engineering)Mar Athanasius College of Engineering, Kothamangalam,kerala, India. Born on 27 th Nov 1991.He has completed his graduation in Mechanical Engineering, from Annamalai University, Chidambaram.

Dr Benny Paul , Professor, Department of Mechanical Engg, Mar Athanasius College of Engineering, Kothamangalam

ISSN: 2278 – 7798