CFD ANALYSIS OF GAS FLOW BEHAVIOR IN VARIOUS ECONOMIZER DUCT

A.Aravindkumar,
Mechanical Engineering,
MookambigaiCollege of Engineering,
Trichy, TamilNadu.
Email: alaravind.ak@gmail.com

Abstract - Efficiency and Energy saving are the key problems in power generation system from the view point of fuel consumption, but also for prevent the environment of the Earth. In the power plant the gas duct is the major part. The gas duct is for release the flues gas which is produced by the boiler. In this project we will design the gas duct and also analysis the duct. The main objective of the project is CFD analysing the new economizer duct. The analysis result is compared with the existing design. The vanes are used to distribute the gas equally from the economizer. The effective heat transfer is done by reducing tubes number in the existing economizer. The economizer design is designed by using SOLIDWORKS software and the analysis is made by ANSYS software.

Keywords - Gas Duct, Computational Fluid Dynamics (CFD), ANSYS Software.

1. Introduction

Economizers modulate outside air intake to minimize/eliminate mechanical cooling in air-handling units when outside air enthalpy is lower than the return air enthalpy. A theoretical study is conducted to developed analytical economizer models, to investigate potential energy savings, and to develop a potential energy savings calculation method for dual-duct air-handling unit systems. The study found that economizers decrease cooling energy consumption without heating energy penalties for dual-fan, dual-duct air-handling units. Economizers have significant heating energy penalties for single-fan, dual-duct air-handling units.The term economizer is used for other purposes as well. Economizers are mechanical devices intended to reduce energy consumption, or to perform useful function such as preheating a fluid.

2.Literature review

Niyati Patil

Heat Recovery Steam Generator economizer duct is the major part, by using the economiser duct relieve the exhaust gas produced from the boiler. The main objective of this work is to get hundred percentage uniform flows of flue gases.By using the ICEM software the economiser duct is developed and analysed. There is number of plates added in the economiser. It is used for used pressure drop and decrease of high velocity of the fluid flow. The main work in this project is increase the uniform flow.

AkramGhouse

When the flow inters through Regenerative Air duct and boiler duct and flows through the exhaust part of the ductsVibrations are observed in the ducts. The boiler ducts are failure in their operation and had increase cracks on layers and damaged the full system by the vibration. CFD analysis is a one of good technique by using we can give solution for the practical for exp in a with the help of which a lot of time, cost and materials can be saved andnumerical manner. Here in this project CFD analysis can be used to predict the flow character of the ducts by the help of which a prediction can be made so as to optimize the design of the Waste heat Boiler duct for minimum vibration.

Indrusiak

The actual operation condition of a thermal power plant, where the water seal in the below of the boiler presented angas leakage, was optimization using the CFD code CFX. The conclusion were compared with the ones for the situation where the leakage does not exist and the full amount of air need for the combustion would be furnished by the secondary gas feeds. The operation condition affects the the flow dynamics of the boiler and also the NOx formation and its performance.

Ramesh Avvari

Flue gas ducting in a power plant encounters erosion at vanes, duct walls frequently due to impact of high velocity ash particles over a prolonged period of the process. This could lead to puncturing of duct walls and vanes often necessitating maintenance or changing, which is normally done during yearly shut downs. Rules for erosion prediction through CFD modelling areimproved for duct walls. This able to the designer to take corrective actions for changing the size of duct to make the flow streamlined and decrease the chances of erosion. Better-streamlined flow in a duct willdecreasepower consumption, pressure drop by fan and maintenance money due to erosion.

3. Problem Definition

 To perform the steady state CFD analysis for waste heat from the boiler ducts with various geometric. The optimization is made in ANSYS 15.0 and is based on governing equation of Mass. Momentum, energy and turbulence. k-ε turbulence model is used for this assessment due to large applications in the flows. To design a suitable dimension for the waste heat from the boiler duct which tend to suppress the turbulence and assigns the flow properly which intern decrease the vibrations. To compare various cases of Waste heat from the boiler with vanes attached to the WHB in various angles and various numbers.

4. Specification

The base model of the Waste heat from the boiler is shown in the figure, based on the design the model is designed, fine meshed and analyzed. Diameter of RA and GT are 2252mm and 1949 mm and Height of duct is 5183 mm respectively. Based on the final result of the present model it was decided that guide plates has to be introduced in the duct. Various cases with various counts and orientation of guide plates are known.
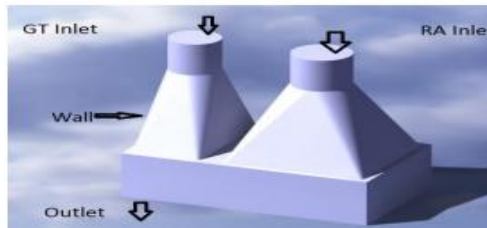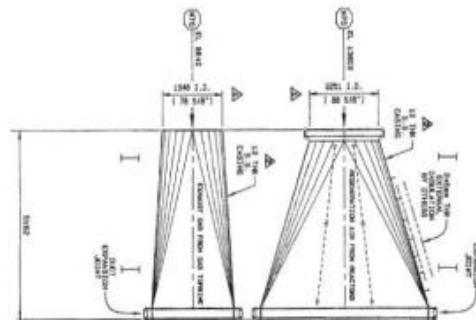
Fig 1 Basic solid model

Fig 2 Geometry

5. Methodology

The dimension of the waste heat from the boiler duct and plates are designed using SOLIDWORKS software. The model with various kinds of plates is analysing by usingANSYS software. The designed model is prepared and analysing using SOLIDWORKS and ANSYS software. Basic 3D model used is as with dimensions 4 guide plates are used, in the CaseII two guide plates are fixed in there, in the Case III four number of guide plates are fixed and finally in the CaseIV4 guide plates are fixed in there then analysis is done.

6. Results and Discussions

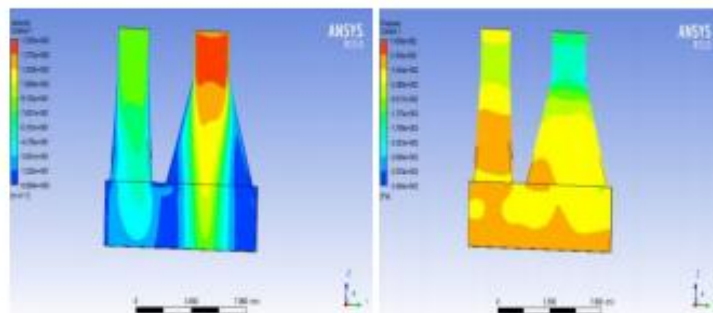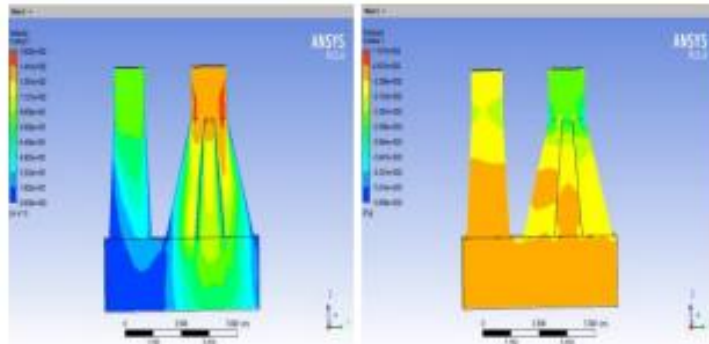**Case I**

Fig 3 & 4Velocity counter and pressure counter

Case II

Fig 5 & 6 Velocity counter and pressure counter
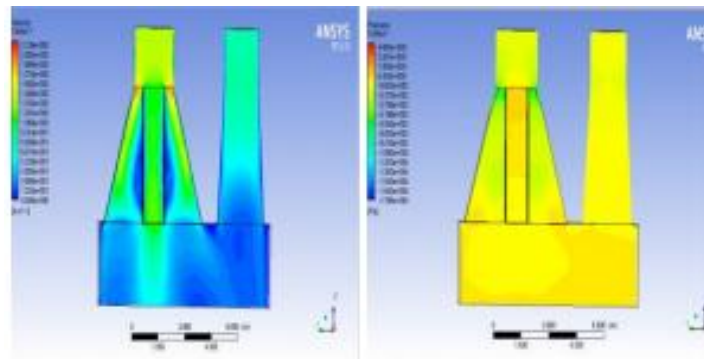
Case III

Fig 7& 8 Velocity counter and pressure counter
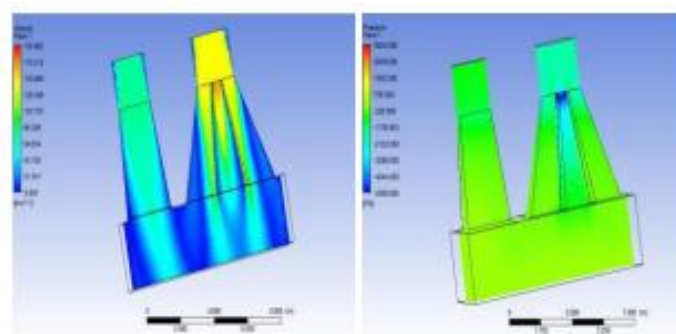
Case IV

Fig 9 & 10 Velocity counter and pressure counter

7. Conclusion

By analysing the four various cases by using the ANSYS software we will choose the best one. The fourth case has more good and accurate flow distribution. And also the fourth case does not had a vibration.From the fourth case result we known that less pressure drop and

less loss of energy and also it has a good uniform flow rate. By compare the four various cases, the fourth case more suitable for the power plant.

References

[1] Bezuidenhout, J.J. Yang, Y. and Eksteen , J.J. "Computational fluid dynamic modelling of a waste-heat boiler associated with flash smelting of base metal sulphides".

[2] Hajee Mohamed, M. and Wilson,V.H. "Analysis of Boiler Super Heater Tubes from High Flue Gas Temperature"

[3Bezuidenhout, J.J. Yang,Y. and Eksteen , J.J. "Computational fluid dynamic modelling of a waste-heat boiler associated with flash smelting of base metal sulphides" [4] SwapnaneelSarma and D.H Das "CFD Analysis of Shell and Tube Heat Exchanger using triangular fins for waste heat recovery processes"

[5] Reay, D.A. & Span, F.N. "Heat Recovery Systems" , London, 1979.

[6] Versteeg, H .K. and Malalasekera ,W."An Introduction to Computational Fluid Dynamics THE FINITE VOLUME METHOD Second Edition".

[7] "Analysis of Boiler Super Heater Tubes from High Flue Gas Temperature"