ATTAINMENT OF FULLY DEVELOPED FLOW IN AIR DISTRIBUTION DUCT OF BOILER USING CFD ANALYSIS

P.Devakumaran[1], C.Balakrishnan[2], S.Chandran[3], C.Azhagurajan[4], B.Veluchamy[5], M.Vivekanandan[6]

[1,2,3,4] *UG Students, Department of Mechanical Engineering, K.Ramakrishnan College of Engineering, Tiruchirappalli, Tamilnadu, India.*

[5] *Asst Prof, Department of Mechanical Engineering, K.Ramakrishnan College of Engineering, Tiruchirappalli, Tamilnadu, India.*

[6] *Chief Executive Officer, TryCAE Industrial Engineering, Tiruchirappalli, Tamilnadu, India.*

ABSTRACT

In the power plant the gas duct is the major part of the system. In this project the gas duct is used to release the flue gas from the boiler outlet to the economizer input port. For modeling the duct SOLIDWORKS software is used. Here in this project CFD analysis can be used to predict the flow behavior and flow of the gas distributed equally at inside of the ducts. The main objective of this project is analyzing the gas duct for even distribution of flow. For even distribution the vanes are used. Totally four cases are tried which consists of a base case without the guide plates and three cases with guide plates are tried. The solution is to get even flow distribution of hot gases inside the duct .Duct model was modified by modeling number of guide plates and angles to get distributed flow of hot gases inside the duct. Finally compare the existing design with modified new design.

Keyword: *Solid works, CFD (Computational Fluid Dynamics), Duct, Vanes.*

1. INTRODUCTION

In power plant the duct can be used to connect the one system to the other system. The main purpose of ducting system is transfer the flue gas from the boiler to the other systems like economizer and air pre heater. By using the duct in power plant we can reduce the pressure drop and equal flow of the gas to the other systems. The duct can be made by using the Galvanized steel, aluminum, and Polyurethane and phenol insulation panels, fiberglass duct board. As per the study found that the dual ducting system can be used to separate the flue gas from the boiler and attain the even distributed flow for the both economizer and air pre heater by using the vanes (flow modifier).

2. LITERATURE REVIEW

MADHULIKA SINGH, SHAH ALAM works on "CFD Approach to Design and Optimization of Air, Flue Gas Ducting System" In this paper focused on duct system is used to transfer of air or gas from one place to another place and chamfering the required section to have the uniform distribution of the flue gas through the duct. And also pre circulate the gas before entered into the duct. The main objective of this paper is by using the CFD analysis to

provide the improved flow condition in air and flue gas in the duct and also to reduce the pressure loss and low turbulence

A.ARAVINDKUMAR, works on "Analyzing the gas flow in CFD for various ducts". In this paper focused on to form a vibration less duct and good flow distribution the number of plates provided in the inlet of the ducting system. There are four cases were studied in this project. The number of plates has increased in each case for reduce the vibration and also provide the even distribution of flow. The main objective of this paper is analyzing the new economizer in CFD. Finally the existing design is compared with the new modified design.

SREEKANTH S, Dr. BENNY PAUL, works on "CFD analysis of waste heat boiler". In this paper focused on 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. The vibration developed was reduced the Turbulent flow of the gas. So the suitable solution was found that the guide plats are introduced to reduce the vibration. And also reduce the cost of the process.

J.J. Bezuidenhout, Y. Yang and J.J. Eksteen has investigated waste heat boiler in which the waste-heat boiler is used within the supplied flash smelting process as the main dust and energy recovery unit. Computational fluid dynamics (CFD) is applied within a study to model the flow and heat transfer distribution throughout the waste-heat boiler. This study focuses on the geometric modifications to the boiler, which includes elevation of the ceiling, placement of flow- obstructing baffles and radiation plates parallel within the flow path. The geometric modifications had the desired effect of increasing the volumetric utilization and therefore enhancing heat transfer between the boiler surface and the gas stream and dust segregation.

3. OBJECTIVE OF THE PROJECT WORK

1. To equally distribute the flow of hot gas into specified ducts.
2. To analyze the attainment of fully developed flow in air distribution duct of boiler using **CFD.**
3. To attain efficiency and energy saving in power generation system for fuel consumption and prevent environmental issues.
4. Check whether the flow of the gas distributed evenly to the both ducts called dual ducting system in according to the vanes placed.
5. To compare the modified design analysis result with existing design analysis Results.

4. PROBLEM IDENTIFICATION

1. The distribution of the hot flue gas was varied for the both duct with respect to the amount of the flow and velocity of the gas.
2. The efficiency of the power plant system was reduced.
3. The power consumption due to the uneven flow the gas has increased and the environmental problems has also increased.
4. The recirculation of the gas after came out from the economizer amount is varied.

5. METHODOLOGY

1. The geometry of the dual ducting system was designed by using the SOLIDWORKS software.
2. The model without the flow modifier (guide plates or vanes) flow was analyzed by using the CFD software and the values are noted.
3. The mesh also done by flow simulation software.
4. In model 1 i.e., the geometry with guide plates at the chamfered places are analyzed and results are taken.
5. following that model the model 2 guide plate at the inlet, model 3 guide plate at the dual duct forming place and model 4 combined all the model results are tak

6. BOUNDARY CONDITIONS

Type	Inlet volume flow
Faces	Face<1>@LID3
Coordinate System	Face coordinate system
Reference axis	x
Flow parameters	Volume flow rate : 5.2500m^3/s Fully developed flow : Yes
Thermodynamic parameters	Pressure : 101325.00pa Temperature : 303.00 K

7. RESULTS AND CONCLUSION

The below mentioned model is the initial design of the dual ducting system is created by using the SOLIDWORKS software. This model is designed by without using the flow modifier or vanes or plates.

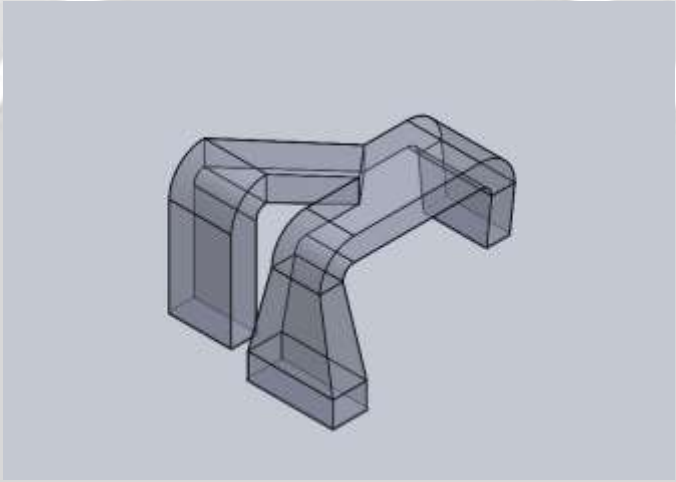

Fig 7.1 Initial design without analysis

The model without the analysis results is obtained by following. It shows the uneven flow of the hot flue of the gas come out from the boiler.

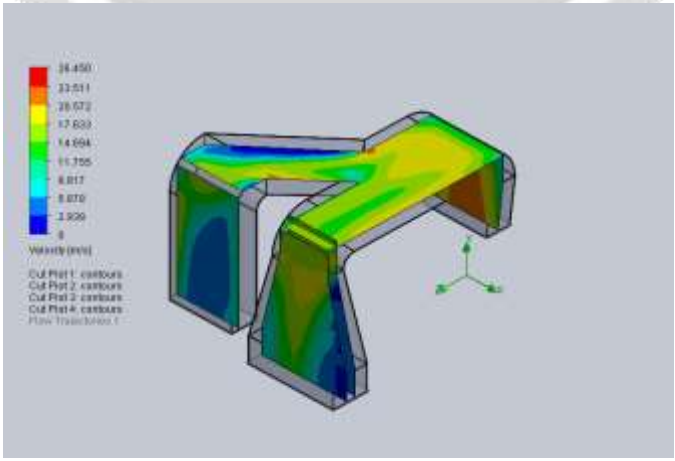

Fig 7.2Without vanes analysis

Dual ducting system analysis with the plates provided at the chamfered place only. It shows the slight flow variation than the previous initial design.

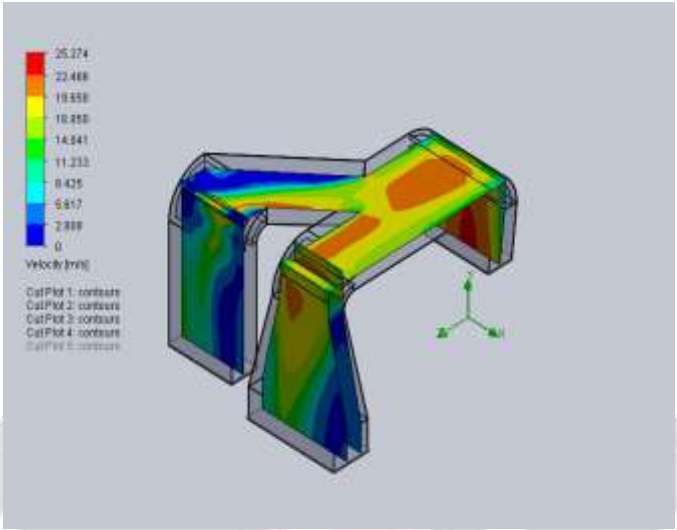

Fig 7.3 Vanes at chamfered place analysis

Dual ducting system analysis with the plates provided at the chamfered place and also the inlet of the dual duct, the outlet of the both ducts, dual duct splitting places are the vanes provides here. It will give the fully developed flow the dual duct for getting a better efficiency.

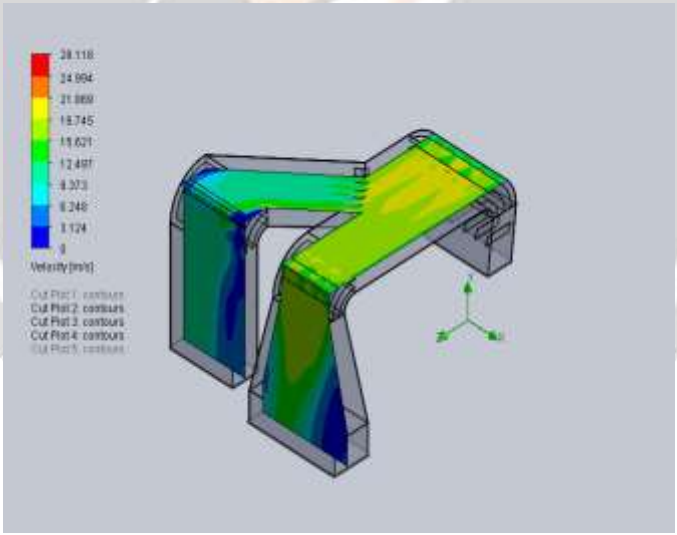

Fig 7.4 Vanes at inlet, outlet of the system analysis

The final analysis consists of the plates at all the mentioned places like the chamfered place, inlet of the duct and outlet, duct splitting place. In the final analysis the objective of the project is achieved by getting the equal flow at the both the outlet of the dual ducts.

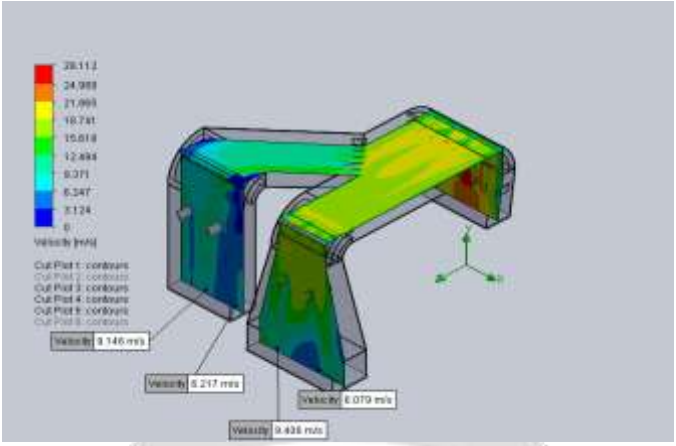

Fig 7.5 Vanes at all the places (final analysis)

8. CONCLUSION

Based on the findings obtained, the following significant conclusions can be made:

1. Computational Fluid Dynamics of the given design without guide plates and with guide plates were carried out to check for the fully developed of air.
2. The placement of guide plates will provides the better fully developed flow of air through the duct when compared to the without guide plates.
3. By analyzing the four various cases the fourth case guide plates at all places will provide the better flow for us.

9. REFERENCES

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

[2] Krunal P. Mudafale & Hemant S. Farkade "Simulation of economizer zone model using CFD techniques by STAR-CCM+software in which the individual tudes are treated as sub-grid features.

[3]. Mautbetsch J., Offen G., "Guidelines for the Fluid Dynamics Design of power plant Ducts," Electric Power Research Institute California, 1988.

[4]. Neihad AL-Khalidy, "Design Optimization of Industrial ducts using computational fluid dynamics," Third International Conference on CFD in minerals and process industries, CSIRO Melbourne Australia, 2003.

[5]. Menke, Michael M. (1994), Improving CFD Decisions and Execution, Research Technology Management, (Sept- Oct), 25-32.

[6]. S. Jayanti "Techniques for flow passage optimization in air and flue gas ducting in boilers", BHEL National conference on CFD Application ,Hyderabad, 17-18 Nov,2006.

[7] Bezuidenhout, J.J. Yang, Y. and Eksteen , J.J. "CFD Analysis of a waste-heat boiler associated with flash smelting of base metal sulphides".

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

[9] Blake Robert Eisner "CFD simulation with fine and a recenty developed turbulence model, the scale adaptive solution (SAS)model.