Analytical Simulation of Flow Distribution in A Waste Heat Boiler Duct using CFD

Saviour Sampson[1] Dr. M.C Navindgi[2]
[1]M.Tech Scholar [2]Associate Professor
[1,2]Department of Mechanical Engineering
[1,2]P.D.A College of Engineering Kalaburgi-585102

Abstract— Vibrations are observed in the ducts when the flow enters through regenerative air duct and gas turbine duct and flows through the diverging part of the boiler ducts. Due to this vibration, the boiler ducts have failed in their operation and had developed cracks on wall and finally damaged the whole system. CFD analysis is a new technique with the help of which we can solve the practical examples in a numerical manner and with the help of which a lot of time, money and materials can be saved. Here in this project CFD analysis can be used to predict the flow behavior inside 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. After the analysis it is found that CFD has helped in optimizing the model in a way that the introduction of the guiding plates has helped in reducing the disturbance of the flow which in turn will reduce the vibration in the model. Different cases are tried and results and discussion are presented in this report. Totally four cases are tried which consists of a base case without the guide plates and three cases with guide plates are tried. In the second case two guide plates are introduced at the RA duct and results are observed. Two guide plates didn't gave proper flow and hence in third case 4 guide plates in the RA duct were introduced, this option gave better results in the RA duct but the flow in the RA duct the flow was not yet proper. Finally in the fourth case along with four guide plates in the RA duct, two guide plates in the RA duct was made and results were observed. It was found that the fourth model gave best results compared to all cases and hence is an optimized model.

Key words: Waste Heat Boiler, CFD, Ducts, Fluid Mechanics

I. INTRODUCTION

Waste heat is a heat which is generated in a process by combustion of fuel or chemical reaction, this heat is dumped into the atmosphere even though it could still be reused for some other useful economic purpose. Heat is not the amount of heat but rather its usefulness. This heat if not used properly will get waste in the atmosphere and also causes heat pollution in the environment. There is a need to recover this heat to utilize it. Best source for heat is the boilers, kilns, ovens and furnaces. If this heat could be recovered a good amount of primary fuel can be saved. But it is not possible to recover full energy lost through heat.

Industrial waste heat in an energy generated through industrial processes without being put to practical use. It is estimated that as much as 20 to 50% of industrial energy is ultimately discharged as waste heat. The exact quantity is poorly quantified. While some losses are inevitable, in some way we can reduce these losses by improving equipment efficiency or installing recovery technologies. Heat recovery entails capturing and reusing the heat in industrial processes for generating mechanical or electrical energy. Example generating electricity, preheating furnace loads, preheating combustion air, space heating and absorption cooling.

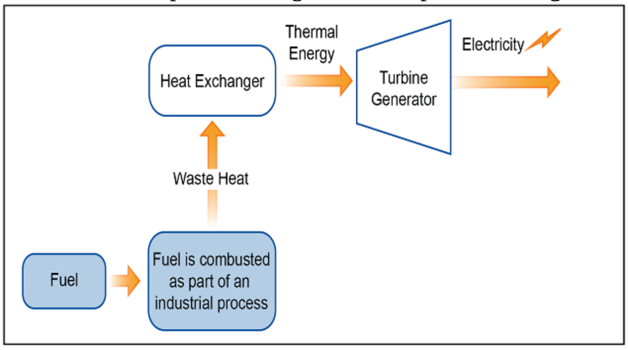

Fig. 1: Flow diagram.

Higher the temperature, higher is the quality and more cost effective is the heat recovery. In any study of waste heat recovery, it is necessary that there should be a use for the recovered heat. Typical examples of use would be preheating of air for gas turbines, space heating, or pre-heating of boiler feed water. An example of waste heat recovery would be where the high temperature stage was used for air pre-heating and the low temperature stage used for process feed water heating or steam rising. Below shows the process of power generation using waste heat.

Numerous researchers have focused on different aspects of the waste heat boiler duct.

J.J. Bezuidenhout, Y. Yang and J.J. Eksteen [1] 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.

Krunal P. Mudafale & Hemant S. Farkade [2] presented a paper on simulation of the economizer zone. The model is solved using conventional CFD techniques by STAR- CCM+ software in which the individual tubes are treated as sub-grid features. A geometrical model is used to describe the multiplicity of heat-exchanging structures and the interconnections among them. This study is a classic example of numerical investigation into the problem of turbulent reacting flows in large scale furnaces employed in thermal power plants for the remediation of ash deposition problems.

Mukesh Rathore , Manoj Kumar Sain , Dr. V. K. Gorana [3] proposed a new cost effective design of diesel exhaust waste heat recovery system (WHRS) for bus rapid transport system (BRTS, Jaipur) and stationary system and proposes a cost effective robust design that meets the principle design constraints for WHRS. Optimization was carried out using CFD software ANSYS FLUENT and ANSYS WB (Parametric Study). Possible waste heat recovery system geometries studied that meets both Indian pollution control standard and geometries constraints. Based on the numerical analysis, author proposed a new kind of WHRS that meets the performance specifications but is outside of the space constraints of BRTS. The WHRS perform at approximately 0.3 to 0.6 efficiency; providing 10 KW at ideal conditions. The overall length of the system is 1200 mm and the overall width is 550 mm. The weight of the heat exchanger is 100 kg; therefore, it does not meet the weight constraint. The overall cost of the WHRS is INR 15000 which is within budget for BRTS, Jaipur.

A. *Objective of The Project Work*

1) To perform the steady state CFD analysis for waste heat boiler ducts with different geometric consideration. The simulation is performed in ANSYS 15.0 and is based on the use of governing equation of Mass. Momentum, energy and turbulence. k-ε turbulence model is used for this assessment due to wide applications in internal flows.
2) To construct a suitable geometry for the waste heat boiler duct which tend to suppress the turbulence and distributes the flow uniformly which in turn reduces the vibrations.
3) To compare different cases of Waste heat boiler with guide vanes attached to the WHB in different angles and different numbers

II. PROBLEM DEFINITION

- To perform the steady state CFD analysis for waste heat boiler ducts with different geometric consideration. The simulation is performed in ANSYS 15.0 and is based on the use of governing equation of mass. Momentum, energy and turbulence. K-ε turbulence model is used for this assessment due to wide applications in internal flows.
- To construct a suitable geometry for the waste heat boiler duct which tend to suppress the turbulence and distributes the flow uniformly which in turn reduces the vibrations.
- To compare different cases of Waste heat boiler with guide vanes attached to the WHB in different angles and different numbers.

III. SPECIFICATION OF WOKING MODEL AND GEOMETRY DOMAIN

Figure 1 shows the base model of the Waste heat boiler, based on the drawing the model is created, meshed and analyzed. Height of duct is 5182 mm and diameter of RA and GT are 2251mm and 1948 mm respectively. Based on the results of the first model it was decided that guide plates has to be introduced in the duct. Different cases with different number and orientation of guide plates are studied.

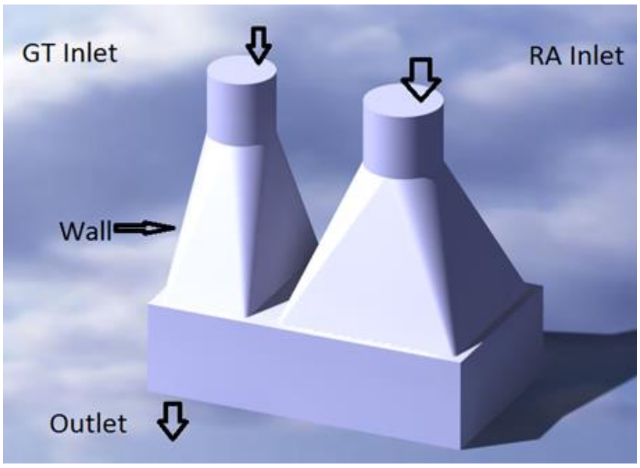

Fig. 2: geometry domain.

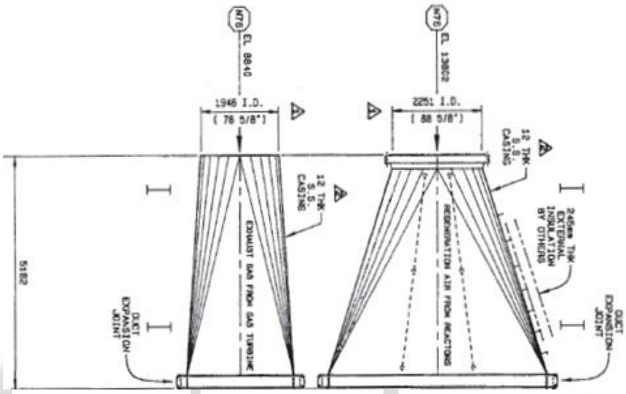

Fig. 3: Geometry of Waste Heat Boiler Duct.

IV. METHODOLOGY AND CFD SIMULATION

The Geometry of the Waste Heat Boiler Duct and Guide Plates are designed Using Computer Aided Drawing. The model with different Guide Plates Are Analyzed Using Commercial CFD Tool. The model is prepared and discretised using CFD-Grid Generation Tool. Basic CAD model used is as shown with all dimensions four guide plates are used, in the model 2 two guide plates are placed in the RA, in the model 3 Four guide plates Are placed in RA and finally in the 4 Four guide plates are placed in the RA with two Guide plates are placed in GT then analysis is done.

Fluid	Flue Gas
Temperature at RA & GT	857 K & 901 K
Mass Flow rate at RA & GT inlets	337 kg/s & 147 kg/s
Density	0.62125 kg/m^3
Viscosity	0.0284 kg/m-s
Conductivity	0.042749 w/m-K
Specific heat	1099.87 J/kg-K

Table 1: Material fluid properties

V. RESULTS AND DISCUSSIONS

The various commercially available tools are used to evaluate the various flow variables at all nodes and the pressure and velocity contour plots are generated are shown.

A. Model 1: Base Model Without Guide Plates:

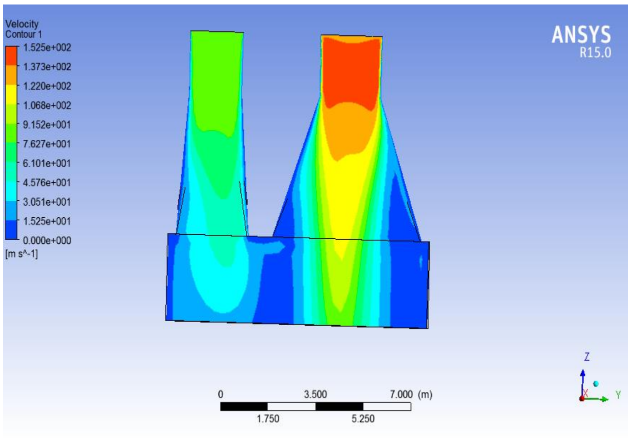

Fig. 4: Velocity Contours

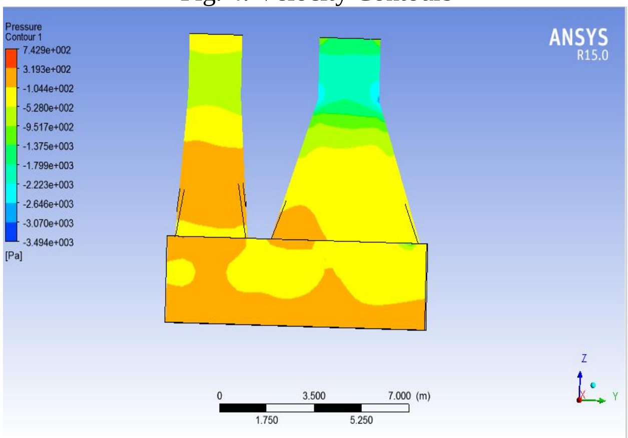

Fig. 5: Pressure Contours

Fig 5.1 and 5.2 shows velocity and pressure contours. It is observed from the streamlines and velocity distribution that flow is not uniform due to one side flow of fluid. But as compared to experimental values of pressure, the pressure drop is are not moving in well defined path, result see that the flow has moved to one part of the model due to which the vibrations will occur. In order to avoid this vibrations guide plates are introduced in the model. Different cases are considered with different number of guide plates and orientation of guide plates is tried and results are shown below.

Velocity contours shows that the distribution of velocity is non-uniform due to non-uniformity of flow of air. In the figure of pressure contour, the pressure is increasing uniformly due to divergence in the model.

B. Model 2: Two Guide Plates Placed In Ra:

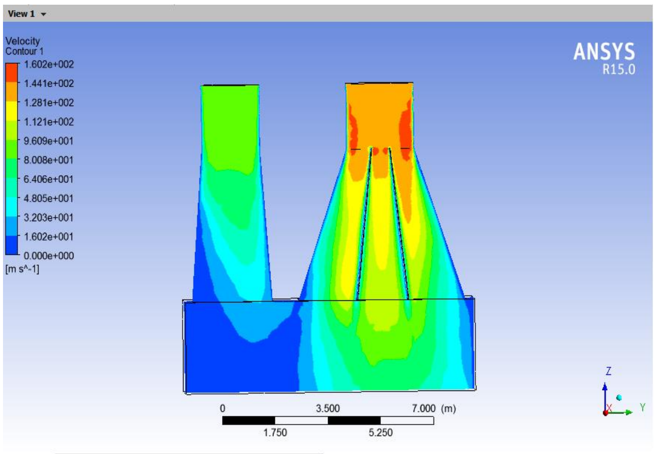

Fig. 5.3: Velocity Contours

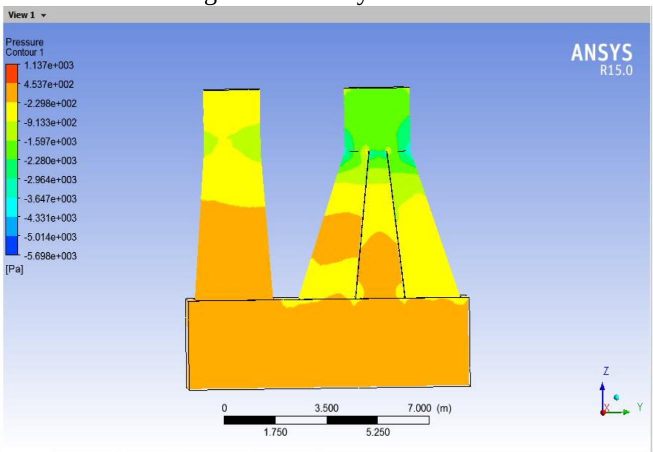

Fig. 5.4: Pressure Contours

Figure 5.3and 5.4 shows the velocity contours and pressure contour. As can be seen that the distribution of flow is better as compared to the previous model. The flow is distributed uniformly in the left and right side of the model in the RA duct, but the flow is not distributed properly overall in the model hence modification is required.

Velocity contours shows that the distribution of velocity is little uniform but not totally uniform.

From the figure of pressure contour, the pressure is increasing uniformly due to divergence in the model.

Another case is tried with plates in different direction and hence results are shown below.

C. Model 3: Four Guide Plates Placed in Ra:

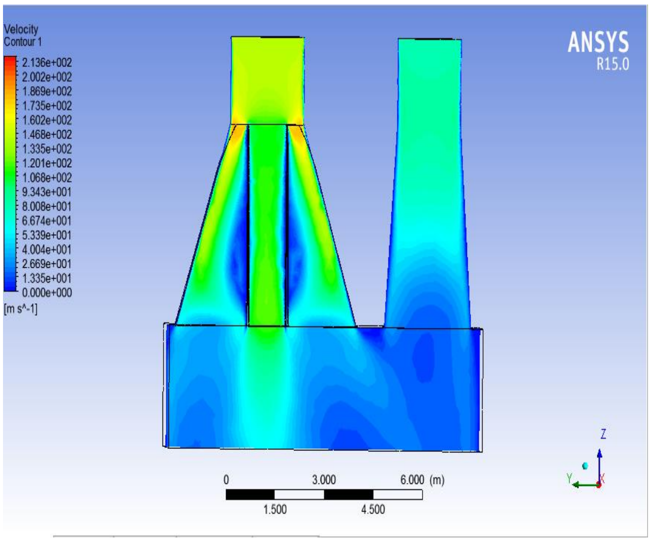

Fig. 5.5: Velocity Contour

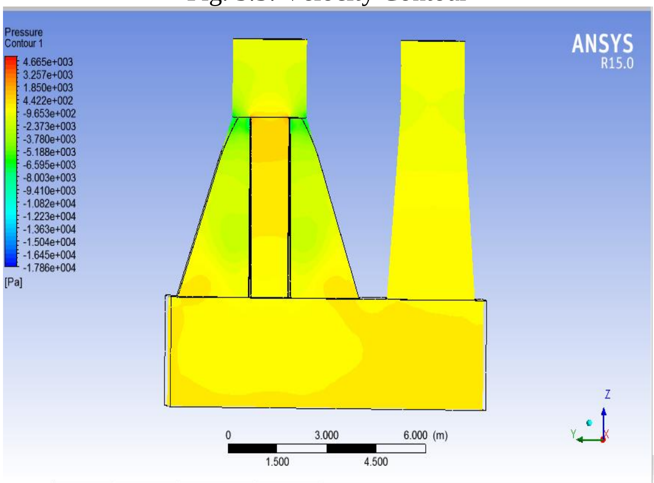

Fig. 5.6: Pressure Contour

From this analysis it is found that the flow of gas is better in the RA duct with flow getting uniform in all the four ways. But the flow is still not uniform in the GT duct which will cause the vibrations in the GT duct and hence it is required to reduce the flow non-uniformity in this duct. One more case with guide plates in the GT duct is introduced and results are observed.

As can be seen from the velocity and pressure contour the distribution of velocity and pressure is uniform in this case compared to previous cases where it was non-uniform.

D. Model 4: Four Guide Plates Placed In Ra And Two In Gt:

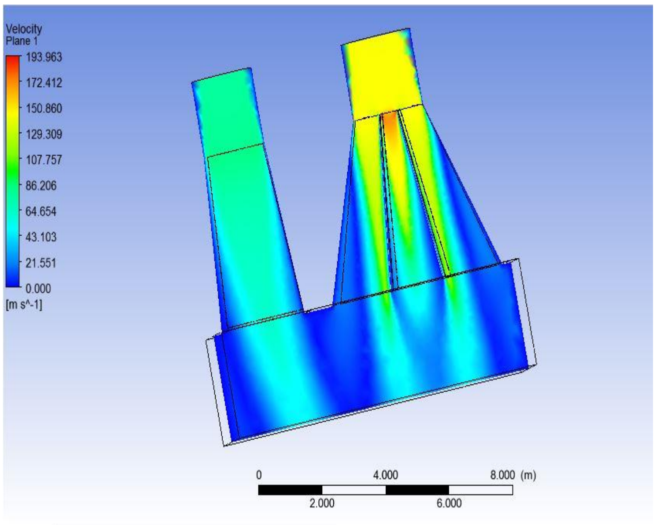

Fig. 5.7: Velocity Distribution Contour

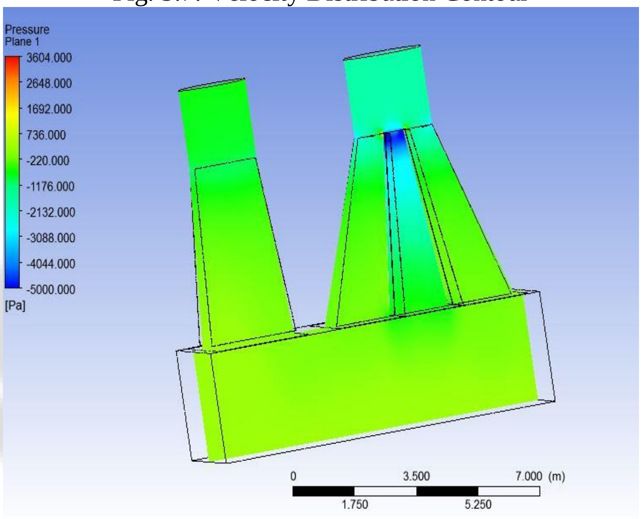

Fig. 5.8: Pressure Distribution Contour

Figure 5.7 and 5.8 shows the velocity contours and pressure contour. It can be seen that the distribution of flow is best compared to all the previous models. The flow is distributed uniformly in the left and right side of the model in both the RA and GT Duct.

From velocity contour it shows that the distribution of velocity is uniform and it gradually decreasing at the exit. In the figure of pressure contour, the pressure is increasing uniformly due to divergence in the model.

After observing this result it is confirm that this model is best for reduction in the flow vibration and hence is considered as optimized shape for Waste Heat Boiler duct.

Case	Total Gauge Pressure at inlet of RA duct(Pa)		Total Gauge Pressure at inlet of GT duct(Pa)		Total gauge Pressure at Outlet(Pa)		ΔP(Pa)
	Experimental	CFD	Experimental	CFD	Experimental	CFD	
Without Plates		3470		1324		410	1987
With Two Side Plates		3341.3		1323.63		600	1732
With four plates in RA Duct.	3259	3481.65	1189	1113.32	398	627	1670
With four plates		3445.85		965.147		655.70	1550

Table 2: Results of Various Parameters.

It can be observed from the results that the fourth model has given the lowest pressure drop since the flow is uniform and hence there is lower loss of energy as the pressure drop is nothing but the loss of energy.

Figure above shows the streamlines, velocity contours and pressure contour. It can be seen that the distribution of flow is best compared to all the previous models. The flow is distributed uniformly in the left and side of model in both RA and GT duct.

From velocity contour it shows that the distribution of velocity is uniform and it gradually decreasing at the exit.

In the figure of pressure contour, the pressure is increasing uniformly due to divergence in model.

VI. Conclusion

In second model the flow distribution is better compared to case 1 but still not satisfactory. Model 3 and 4 are giving better results compared to model 1 and 2. In model 3 the flow distribution at the RA duct is perfect and can be considered as final and also the flow in the GT duct is also better compared to previous cases but there is a possibility of improvement. The pressure drop for third and fourth model is less compared to first two cases. In model 4 the results are perfect since the guide plates are provided in RA and GT duct and hence flow is distributed uniformly in the both the ducts without hitting the duct on one side which in turn will reduce the vibrations. Also the pressure drop in fourth model is less compared to all the three cases. From all the cases considered here it can be concluded that the flow distribution observed in model 4 is best among all the models as the flow distribution is perfect and hence there will be no vibrations. It can also be observed from the results that the fourth model has given the lowest pressure drop since the flow is uniform and hence there is lower loss of energy as the pressure drop is directly proportional to loss of energy.

Acknowledgment

The authors would like to thank to M.C. NAVENDGI, Associate professor, Dept. of mechanical Engineering, PDA College of Engineering Gulbarga. for an outstanding support.

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] Mukesh Rathore , Manoj Kumar Sain , Dr. V. K. Gorana "Optimization was carried out using CFD software Ansys FLUENT and Ansys WB(Parametri study).
[4] Apu Roy, D.H.Das "CFD Analysis of shell and tude heat exchanger using triangular fins for waste heat recovery processes".
[5] Blake Robert Eisner "CFD simulation with fine and a recenty developed turbulence model,the scale adaptive solution(SAS)model.
[6] M. Hajee Mohamed and Vincent.H.Wilson "Analysis of boiler super heater tudes from high flue gas temperature".