Scientific Journal of Impact Factor(SJIF): 3.134
e-ISSN(O): 2348-4470
p-ISSN(P): 2348-6406
International Journal of Advance Engineering and Research Development
Volume 2, Issue 5, May -2015

CFD Approach to Design and Optimization of Air, Flue Gas Ducting System

Madhulika Singh[1], Shah Alam[2]

[1]PG student, M. Tech (Thermal Engineering), AFSET, Maharishi Dayanand University
[2]Associate Professor, Mechanical Engineering, Section, Jamia Millia Islamia New Delhi-25

Abstract : *In this paper the analysis has been done to study the flow distribution, turbulence and flow separation by the flue gas in the ducting system, relating to the duct system efficiency. The main objective of this work is to have value engineering by CFD (by using ANSYS CFX software) by providing improved flow conditions in the air and flue gas ducting system with minimum pressure losses and low turbulence. Also, The purpose of this study to evaluate the CFD Tool (ANSYS CFX); as how it has helped to organization to have improved efficiency by reducing pressure drop in the ducting system and wealth creation (by weight saving).*

Keywords- *ANSYS-CFX, turbulent zones, flow separations, Duct, pressure drop, swirl*

I. INTRODUCTION

Ducting system is used in a power plant to connect one system to another through which cold air, hot air and flue gas is conveyed along with ash. The purpose of duct designing in a power plant is to achieve a good and reliable plant operation by minimizing pressure drop in each component of a duct system i.e. bends/elbows, diffusers, flow junctions, manifolds, stack inlets, fan inlets and outlets, internal supports, dampers, and expansion joints. The system pressure loss usually varies between 40 and 80 inches of water (between 10 and 20 kPa) depending upon what pollution control equipment is used and how the equipment and ducts are designed to handle the gas flow. [1] Neihad Al-Khalidy [2] had modified the design of various industrial ducts using CFD, Yakhot and Orszag [3] had studied the flow model using Isothermal *k*-Epsilon turbulence model with scalable wall function. The analysis has been carried out for 100% design fuel at boiler MCR condition with available process parameters. All the new design layouts were released without changing the matching sections. J.L. Ferrín, L. Saavedra [4] had developed CFD model for pulverized mill duct systems, Aslam Bhutta et. al [5] has been employed CFD for analyzing pressure drop in various heat exchangers.

In the present study the preliminary layouts for ducting were studied. A 3D-model was generated using ANSYS Design Modular. The complete flue gas ducting was divided into various sections according to the flow path of the flue gas. The model has been incorporated into commercial CFD code using ANSYS-CFX. The flow of flue gas distribution was studied in all sections of flue gas duct, high turbulent zones and flow separations are identified and accordingly the modifications are proposed. The heating surface in flow paths like Air Heater, Economizer is realized as porous media; required modifications were done to reduce the flow turbulence due to higher velocity and the pressure drop due to swirls. Iteratively the baffles were added to have uniform distribution inside the duct. The recommended modification leads to significant improvement and efficient ducting layout with reduced pressure drop & savings in material weight.

II. BACKGROUND

In times of a more competitive market for the power industry, the striving for short-term profits on behalf of the long term investments is unmistakable. This destructive tendency is evident when the long-term advanced engineering process, as a part of the product development process, is studied. Those companies that can master the act of balancing shot-term and long-term interest in their development process have the perquisites to survive and to be successful in an even more competitive market.

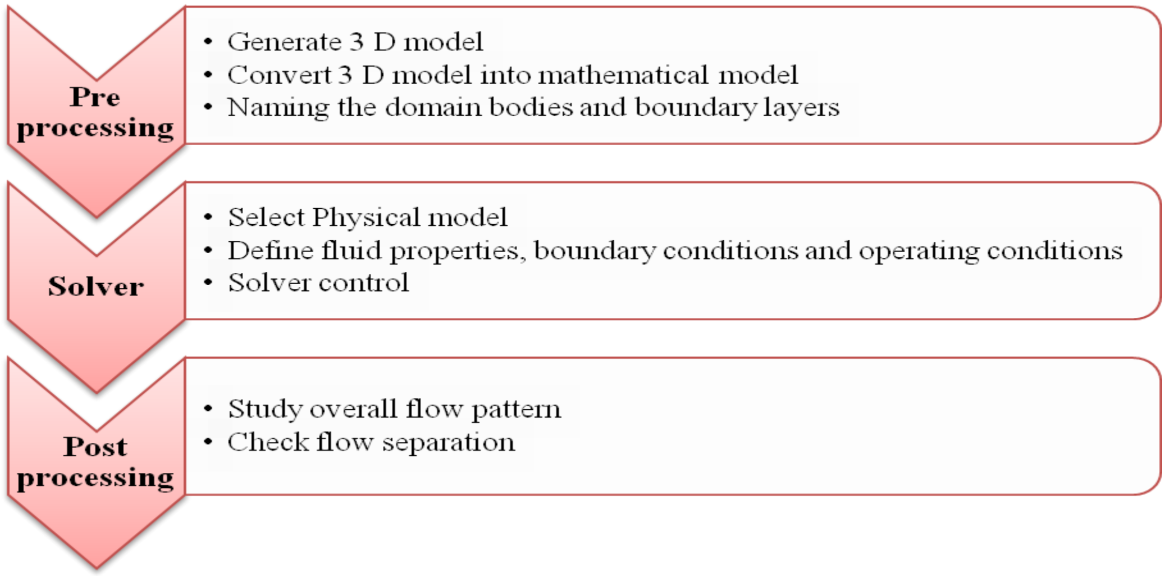
The process of value engineering activities has seen several changes in approach in the last few years and several new management methods have been adopted by industrial houses. In today's scenario, the volatility in price and demand of engineered items such as steel adds to the complexity and builds up further business pressure.

In this backdrop, the significance of this study is very critical. However, the approach has been going through several stages since wealth creation & innovation both need to go hand in hand.

III. PROBLEM IDENTIFICATION AND METHODOLOGY

Decisions made during the layout of a duct system may affect factors such as initial cost, structural behavior, system reliability and maintenance. The design of large air and gas ducts is an extremely complex task that requires the satisfaction of performance, mechanical and structural design criteria. It has to be remembered that the purpose of ducts is to efficiently convey the air or flue-gas from one location to another while, as much as possible, maintaining the temperature and pressure. Flow considerations, such as minimizing pressure drop and controlling ash deposition, are also essential to a good overall design. Velocities in ducts are limited by noise, excessive pressure drop and erosion.

The result of this report is based on a case study. A case study includes studies of a small number of objects in light of many different aspects, in contract to a statistical method, where many objects are studied out of a small number of aspects.

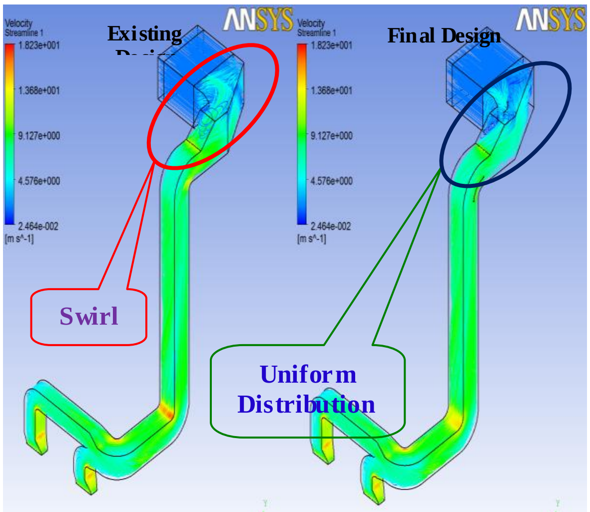


"Figure1. Analysis steps in ANSYS CFX"

Since the aim of this project is to analyze and optimize the ducting system so as to have better ducting efficiency and material saving. For the project three projects ducting layout were taken under consideration. The selection of the projects was done randomly. The project from each boiler type was selected i.e. from CFBC and TG Boiler. Every projects ducting layout was studied and examined for high turbulent zones, re-circulation zones, dead flow zones etc. All these factors are responsible for the losses occurring in the ducting system. After the identification of all the losses, required modification were made in the layout to have material saving in the ducting system with enhancing the ducting performance.

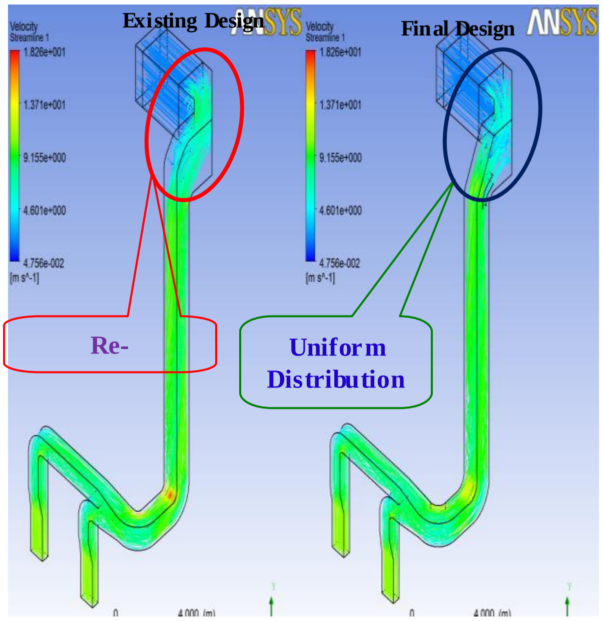
After the modification of the duct weight comparison had been done between the estimated weight and the final weight released in the BaaN. The drawings were referred from either PLM or from the lead engineer.

IV. ANALYSIS AND OBSERVATIONS



"Figure 2a. FD fan to air heater inlet duct"

Observation: The above domain represents FD Fan Outlet to Air Heater Inlet Duct. The streamlines shows that there is swirl at air heater inlet, this leads to loss of KE. To avoid this we have chamfered the required section of the duct. Also the increase in the inner radius of the duct bends to decrease the local velocities at bends.

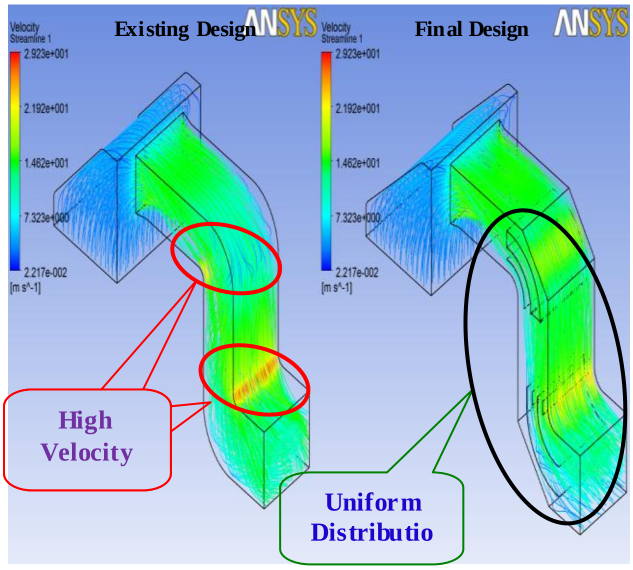


"Figure 2b. SA fan to air heater inlet duct"

The above domain represents SA fan Outlet to Air Heater Inlet Ducting.

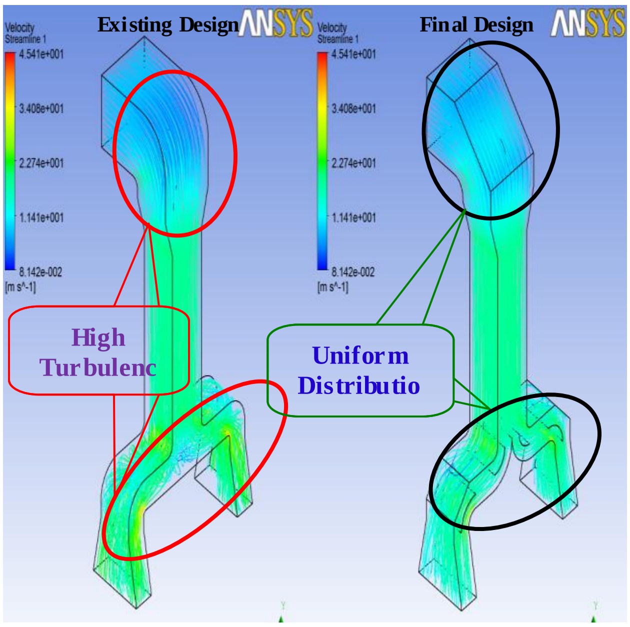
Observation: The streamlines shows that there is re-circulation just before the entry of Air Heater.

This might result in unequal heat transfer, hence affecting the efficiency. It might also damage the air heater tubes.



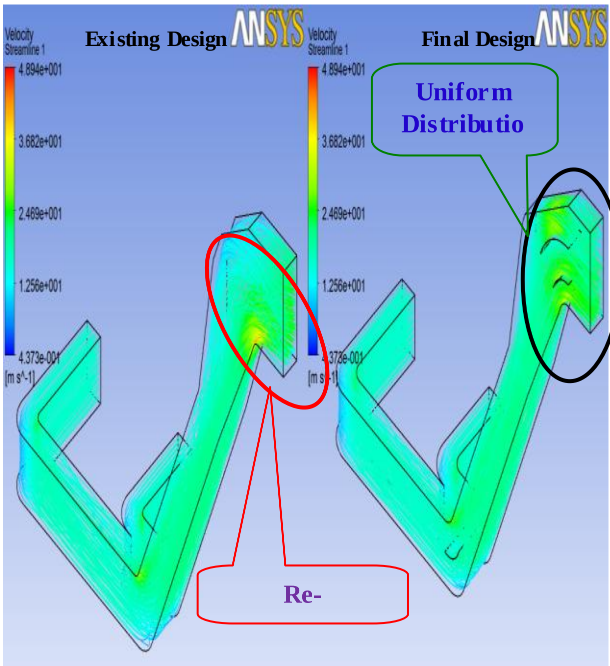
"Figure 2c. Air heater to ESP inlet duct"

Observation: The above domain represents the streamlines in Air Heater Outlet to ESP Inlet Duct. We have chamfered the required section of the duct to have uniform distribution of gas throughout the duct.



"Figure 2d. ESP to ID fan Inlet duct"

Observation: The above domain represents ESP outlet to ID Fan Inlet Duct. We have chamfered the required section of the duct to have uniform distribution of gas throughout the duct.



"Figure 2e. ID fan to chimney duct"

Observation: We have chamfered the required section of the duct to have uniform distribution of gas throughout the duct. The above domain represents ID Fan Outlet to Chimney Inlet Duct.

"Table 1.0 Distribution of pressure drop "

Sl. No.	Description	Pressure Drop (mmwc)		Pressure drop saved
		Existing Design	Proposed Design	
1	PA Ducting - Cold Air	25.83	22.23	3.60
2	SA Ducting - Cold Air	20.52	15.20	5.33
3	AH to ESP Inlet -Flue Gas	3.83	2.58	1.25
4	ESP to ID Fan Inlet-Flue Gas	14	11.59	2.41
5	ID Fan to Chimney - Flue Gas	11.4	9.54	1.86
	Total	***75.58***	***61.14***	***14.45***

V. CONCLUSIONS

The analysis was carried out for all the flue gas ducting; the flue gas ducting is divided into various domains according to the flow path of flue gas. In all domains the flow of flue gas was studied, all the high turbulent zones were identified and accordingly the modifications are proposed.

After the modifications have been made the in the 3D geometry, a 2D sketch for the same is extracted from *Solidworks* and the modified layout of each section of duct is handed over to the lead engineer.

On the basis of the 2D sketch layout, the lead engineer would create all his duct drawings and supports drawing for the duct. Thus would update the same drawings in PLM. The final drawings in PLM would have the final weight of the ducting. The same would be updated in BaaN.

BaaN would have separate weight for each section, namely SA Air ducting, PA air Ducting and Flue gas ducting. The final weight entered separately in their corresponding element in BaaN.

Limitations

The conventional approach to strategy has emphasized setting goals and developing the means to achieve them by matching the resources of the firm (strengths and weaknesses) with opportunities and threats in the external environment, which includes, especially, customers and competitors, and deciding which industries, businesses, or product-market segments to compete in.

REFERENCES

[1]. Mautbetsch J., Offen G., "Guidelines for the Fluid Dynamics Design of power plant Ducts," Electric Power Research Institute California, 1988.
[2]. 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.
[3]. Yakhot V., Orszag S., Thangam S., Gatski T., Speziale C., "Development of turbulence models for shear flows by a double expansion techniques," Phys. Fluids A Volume 4, No7, 1992.
[4]. J.L. Ferrín, L. Saavedra , "Distribution of the coal flow in the mill-duct system of the As Pontes Power Plant using CFD modeling," Fuel Processing Technology, volume 106, Pages 84-94, February 2013.
[5]. Bhutta M. M. A., Nasir H., Bashir M. H., Khan A. R., Ahmad K. N., Khan S., "CFD applications in various heat exchangers design": A review Article, Applied Thermal Engineering, volume 32, Pages 1-12, January 2012.
[6]. Allio, Robert J. and Desmond Sheehan , "Allocating CFD Research Effectively," Research Management, Pages 14-20, July-August1984.
[7]. Bachman, Paul W., "The Value of CFD in Relation to Company Profits," Research Management, 15, Pages 58-63,May 1972.
[8]. Menke, Michael M. (1994), Improving CFD Decisions and Execution, Research Technology Management, (Sept-Oct), 25-32.
[9]. 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.