iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 9 **Issue:** VIII **Month of publication:** August 2021

DOI: https://doi.org/10.22214/ijraset.2021.37273

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VIII Aug 2021- Available at www.ijraset.com

CFD Analysis of Duct from APH Outlet to ESP Inlet

Dhananjay Kumar[1], Dr. Siddharth S Chakrabarti[2]
[1, 2] Department of Mechanical Engineering, O.P. Jindal University Punjipathra, Raigarh, Chhatisgarh, India

Abstract: In generation, Most of the thermal power plant is installed ESP with Bag-house FF Filteration system. Electrostatic Precipitator (ESP) are well accepted and widely used for air pollution control due to reasonable collections efficiency, low pressure-drop, and low capital and operating costs. The performance of modern ESP's would be expected to be better than 99.9% particulate removal efficiency while incurring less than 100 Pa pressure drop. The geometry of inlet duct has refined with guide vanes placed at several locations. An improvement of flow distribution in the duct through optimization can be achieved by modification of inlet duct. In this present work, CFD analysis of flow through the APH outlet to ESP inlet duct and ESP Outlet duct is performed by modification of duct , in order to reduce the pressure drop, reduce turbulence as well as achieve uniform distributions among the all streams and minimize erosion of duct walls caused due to high velocity. Reducing erosion of the duct walls results in the reduction of leakages in the ducts. Reduction of pressure drop across the duct saves the power consumption. The ESP inlet and ESP Outlet duct consists of two inlet and two outlets. The simulation studies involving the flow through the ESP inlet and outlet duct are performed using3-D Model development using Solid Works, ANSYS Design Modeler, Grid generation using ICEM CFD (HEXA) and Performing the simulation using ANSYS CFX using structured hexahedral mesh. The pathlines, velocity contours, pressure distributions for various cases involving without modification of duct. The ESP inlet duct and outlet duct modification shows lesser pressure drop with uniform distributions across the duct.

I. INTRODUCTION

An Electro Static Precipitator is a particle control device that uses electrical forces to move the particles out of the flowing gas stream and onto collector plates. A high voltage is applied to the discharge that produces minus ions. The electrically charged dust is accumulated on the collecting electrode by an electrical field.The electrical field that forces the charged particles to the walls comes down electrodes maintained at high voltage in the centre of flow lane as seen in Figure 1. Once the particles are collected on the plates, they must be removed from the plates without re- entering them into gas stream. This is usually accomplished by knocking them loose from the plates, allowing the collected layer of particles to slide down into a hopper from which they are evacuated. Some precipitators remove the particles by intermittent or continuous washing with water. Electrostatic precipitators are used in large power plants, cement plants, incinerators, chemical factories, sugar mills, etc

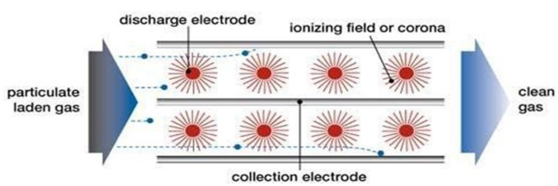


Fig-1

ESP's are well accepted and widely used for air pollution control due to reasonable collection large scale industrial gas cleaning exhibit slight variations in design characteristics, component, and mechanical cleaning efficiency, low pressure- drop, and low capital and operating costs. The overview of an ESP is shown in Figure 2. The performance of modern ESP's would be expected to be better than 99.9% particulate removal efficiency while incurring less than 100 Pa pressure drop. Although commercially available ESP's for style, all achieve gas cleaning quite simply by the use of electrostatic forces acting on airborne solid particles to promote their attraction to and eventual collection on charged electrodes. ESP inlet duct is one through which flue gases enter electrostatic precipitator from air pre-heater. The design of ESP inlet duct is very important that the collection efficiency and the power consumption will be affected. In this present work, the rationale is to increase the collection efficiency of ESP inlet duct and reduce the power consumption of ESP. This is achieved by reducing the turbulence in ESP inlet duct. By reducing the pressure drop across the duct and reducing the turbulence saves the power consumption. By uniform distribution reduces erosion duct walls that result in reduction of leakages in duct. So this is achieved by introducing guide vane in inlet duct.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VIII Aug 2021- Available at www.ijraset.com

Bhasker [1 - 4] carried out flow simulation in inlet ducts with several guide vanes to understand the flow pattern at its exit location. The geometry of inlet duct was refined with guide vanes placed at several locations. The domain of duct geometry around guide vanes were decomposed with several volumes and filled with hexahedral elements. The resulting computational grid was used in fluent solver to predict its flow pattern in the duct. Simulation for the specified conditions predicted uneven flow distribution in the ESP inlet duct. Due to large flow recirculation and turbulent losses in the duct, non-uniform averaged mass flow rates were noticed at duct exit locations. Simulation results suggested that the improvement of flow distribution in the duct through optimization can be tried by placing more guide vanes in the inlet duct. In order to ensure that the results obtained from fluent were meaningful and in right

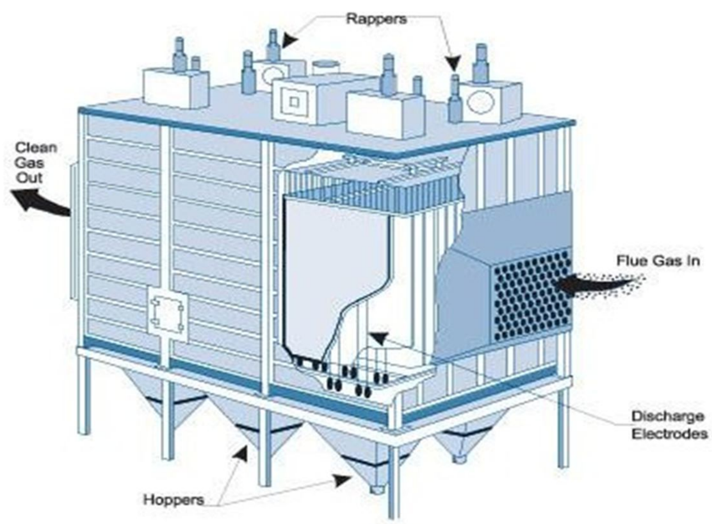


Fig-2

LaRose *et al.* [5] carried out flow simulation of a square duct flow with three-surface-coated multi-layers by RNG k-ε model. It was confirmed that a thicker metallic layer leads to higher pressure loss especially under high Ha numbers while a critical value for Ha/Re to characterize transition from turbulent flow to laminar flow is almost the same regardless of wall conditions under the low Re numbers. Depending on the thickness of the metallic layer, the flow field turns into Hartmann flow or M-shape flow under high Ha numbers. The simulation proved that only the thickness difference of hundreds of μm for the metallic layer was not ignorable in the design of the multi-layer coating, which leads to the necessity of uniform coating technology with high accuracy.

Marian Sarna [6] did the study of particle deposition in turbulent duct flows and comparisons of different model predictions. Numerical studies of transport and deposition of nano and micro-particles in turbulence flow field were done. In most current industrial applications, Reynolds averaged turbulence models were used due to its relative simplicity and computational efficiency. In this work, a series of numerical simulations were conducted to study the transport and deposition of nano and micro-particles in a turbulent duct flow using different turbulence models. FLUENT 6.1.22 was used for turbulence mean flow simulation. Simulations of the instantaneous turbulence fluctuation with and without turbulence near wall correction, and particle trajectory analysis were performed with the in-house PARTICLE (object-oriented C++) code, as well as with FLUENT code with and the use of user's defined subroutines. The simulation results for different cases were compared with the available experimental data, and the accuracy of various approaches was evaluated. In addition, the importance of turbulence model, boundary conditions, and turbulence fluctuation particularly near wall on particle transport and deposition were carefully evaluated. It was shown that when sufficient care was given to the modelling effort, the particle deposition rates could be predicted with reasonable accuracy. The presented results could provide guidelines for selecting appropriate procedure for simulating nano and micro-particle transport and deposition in various applications. The objective of the present numerical investigations is to analyze the flow through the ESP inlet duct. This comparative studies are performed by placing the guide vanes, in order to reduce the pressure drop, reduce turbulence as well as achieve uniform distributions among the streams.

A. Objective

Major Objectives of the project are as follows:

1) CFD Analysis of Existing Ducting System to find out
   a) Uneven gas distribution in four chambers of ESP
   b) High Ash collection in chambers 02 & 03 (Inner side) Compare to Outer side (Chambers 01 & 04)
   c) Recirculation , dead zones & non-uniform velocity areas in the Ducting
   d) High Pressure drop region
   e) Suspected areas of ash accumulations

2) Iterative Modify the Duct by means of Guide-Vane Design & small Duct changes to achieve following Objectives
   a) Mass flow balance in all pass of the ESP within 5% of Avg. Flue Gas Flow
   b) Balance the Ash flow at each ESP inlet within 5% of the Avg. Ash Flow
   c) To achieve uniform velocity in the duct cross section to avoid ash accumulation
   d) To reduce the pressure drop in the Duct to improve the ID Fan margin

B. Boundary Condition

1) Boundary Conditions at Inlet (APH OUTLET)

Flue Gas Flow Rate atAPH -A: 340.44 kg /s
Flue Gas Flow Rate atAPH -B: 324.51 kg /s
Temperature: 120 Deg.C

2) Boundary Conditions at Outlet (ID Fan Inlet)

Outlet-A (ID Fan Inlet-A) : Static pressure : - 3843 pa
Outlet-B (ID Fan Inlet)-B : Static pressure : - 3757 pa

3) Bag House Pressure Drop Considered As follows

Hybrid ESP-A : 140 mmWC
Hybrid ESP-B: 133.39 mmWC
Hybrid ESP-C : 122.79 mmWC
Hybrid ESP-D: 135 mmWC

4) Assumptions

- *Uniform flow at Domain Inlet*
- *Bag House has been Model as Porous Media considering respective Pressure Drop across it*
- *No leakages considered through system*

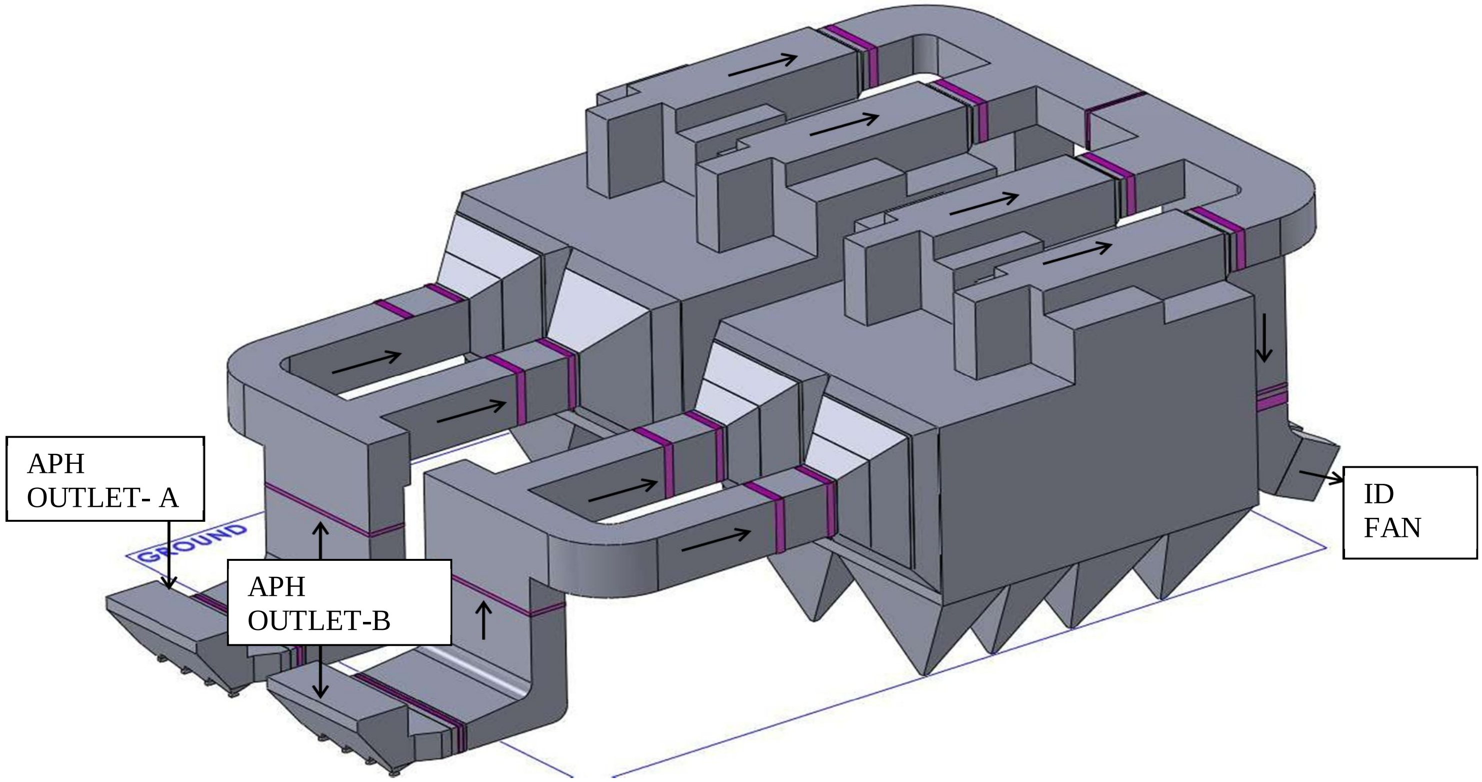
II. METHODOLOGY

The sequence of operations required to carry out the CFD Analysis for the Project as follows:

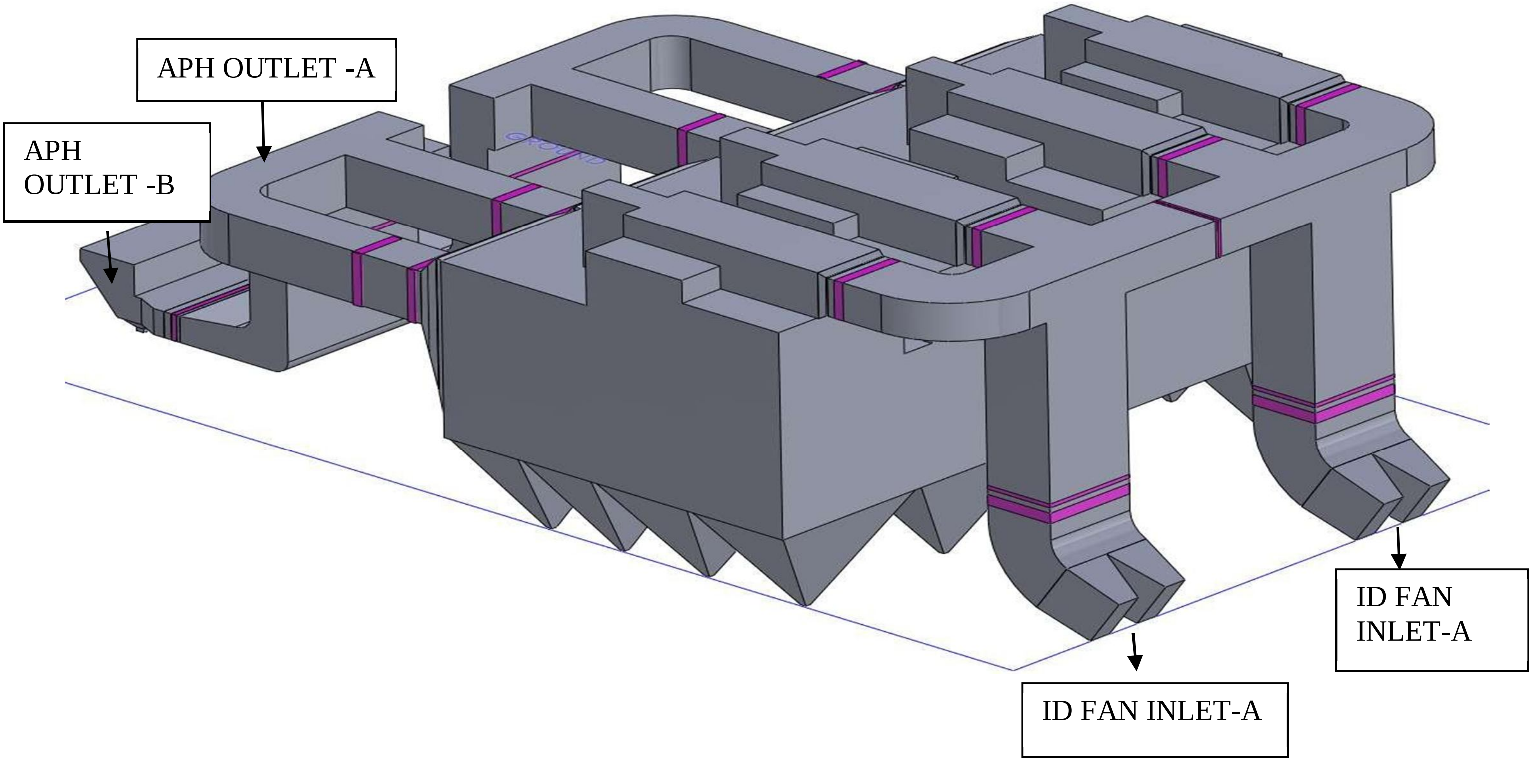
1) Collection of Drawings and Data from client
2) 3-D Model development using Solid Works, ANSYS Design Modeler.
3) Grid generation using ICEM CFD (HEXA).
4) Performing the simulation using ANSYS CFX .
5) Analyzing the results and locating the problem areas in the existing design.

Iteratively modify the design so as to get Optimized Results

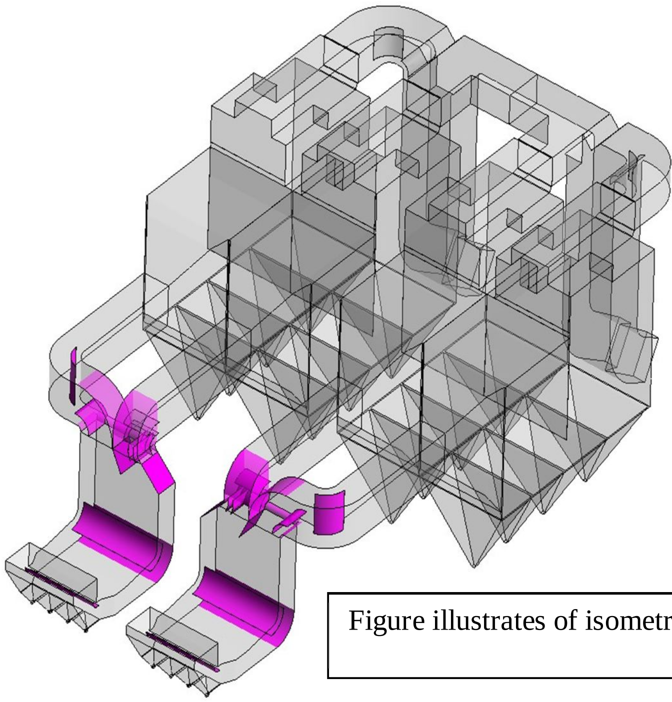
A. 3D Cad Model ESP



B. 3D CAD Model



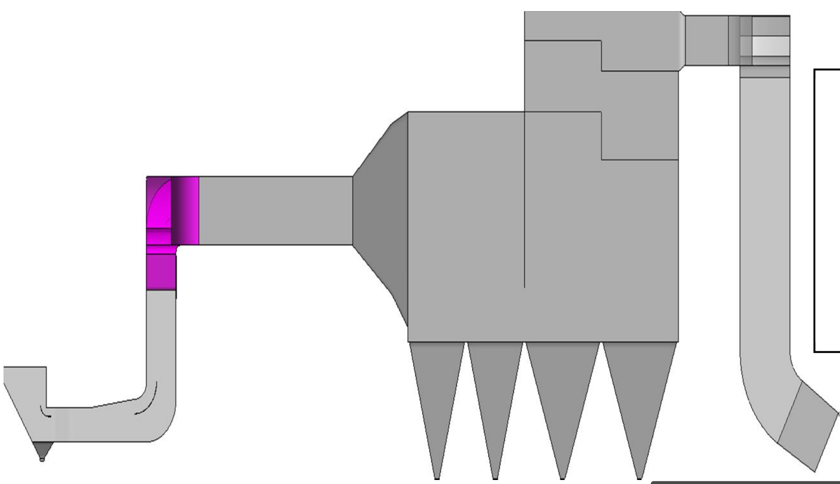
C. 3D CAD Model Modified Duct



- Highlighted portion shows guide plate and duct modification.
- Modifications are conducted to the both pass of the ESP inlet Duct & ESP Outlet Duct

Figure illustrates of isometric view of Modified Duct with ESP

D. 3D CAD Model Modified Duct



- Marked portion shows guide plate modification duct.
- Modification are conducted to the both pass of the ESP inlet duct and ESP outlet duct

Figure illustrates of front view of duct with ESP

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VIII Aug 2021- Available at www.ijraset.com

E. 3D CAD Model Modified Duct

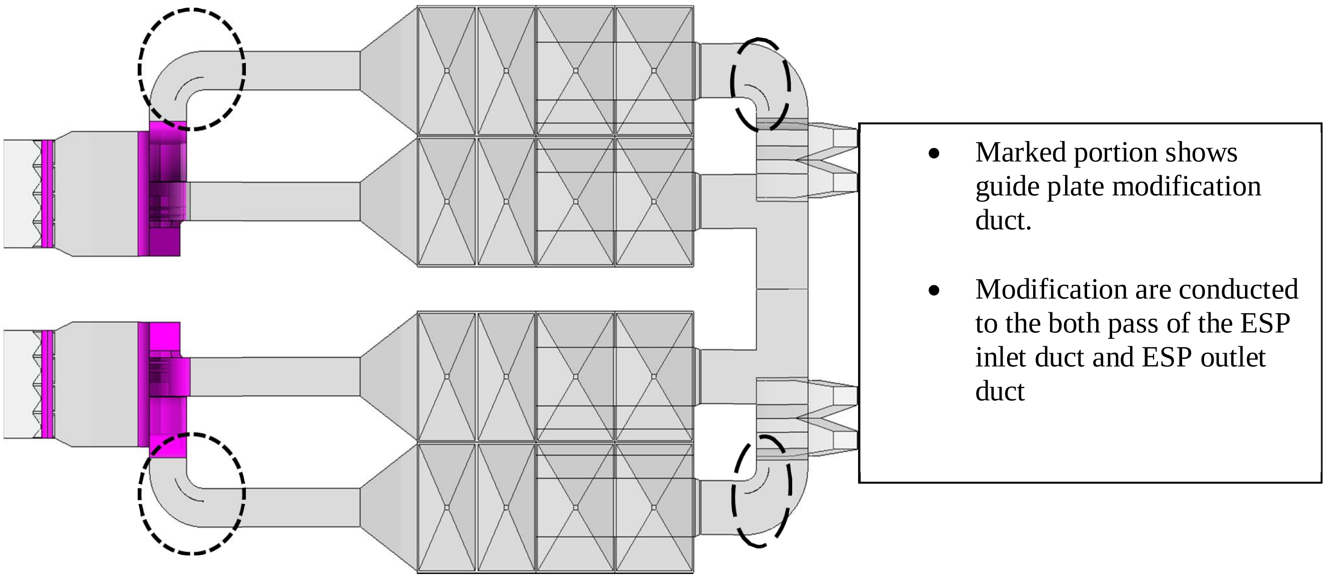


Figure illustrates of isometric view of modified duct with ESP

F. CFD Results Existing Duct

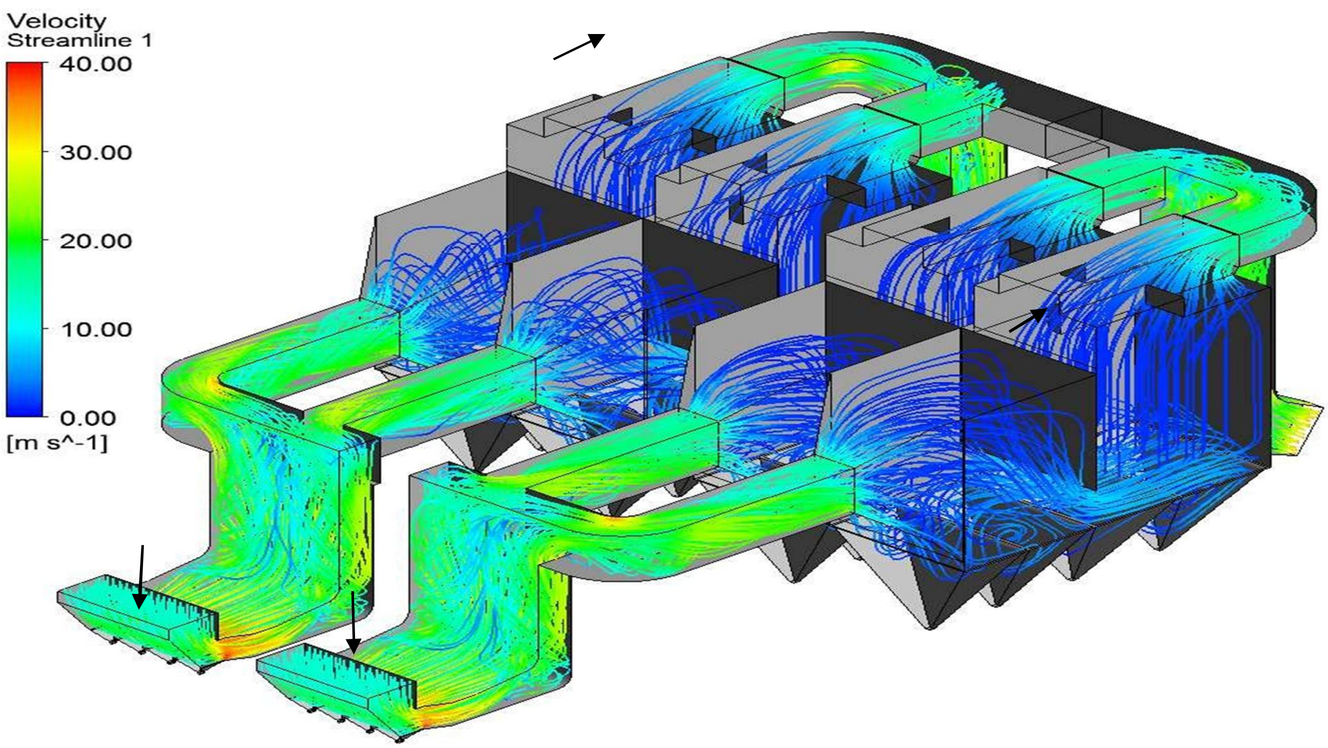


Figure illustrates velocity streamlines for Existing duct

G. CFD Results Modified Duct

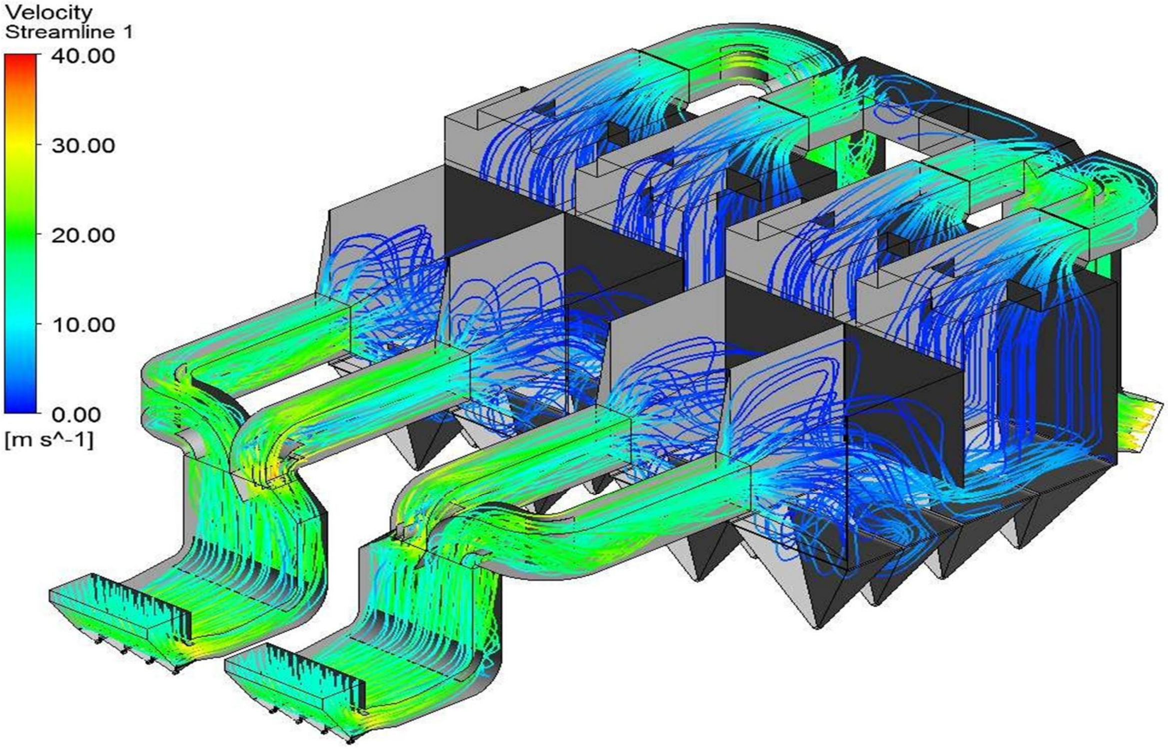
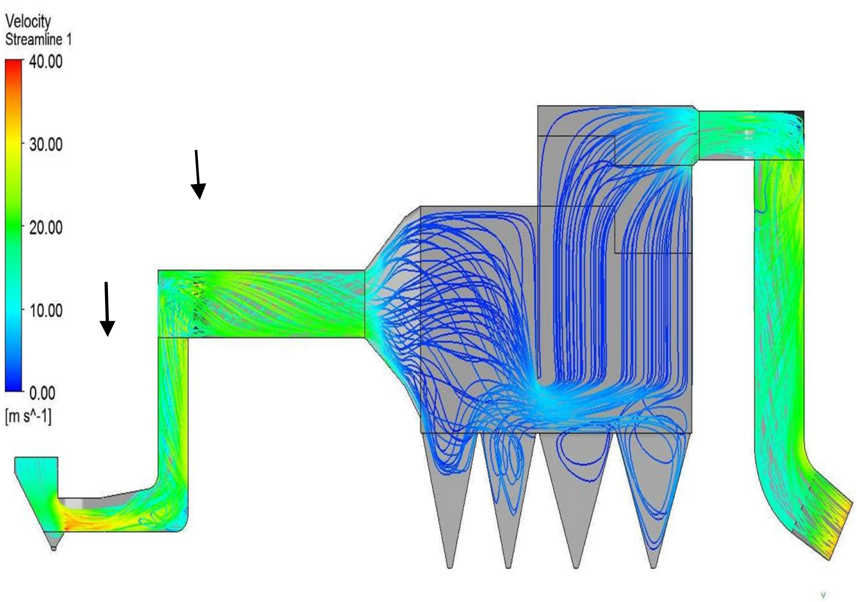


Figure illustrates velocity streamlines for modified duct

H. CFD Results Existing Duct

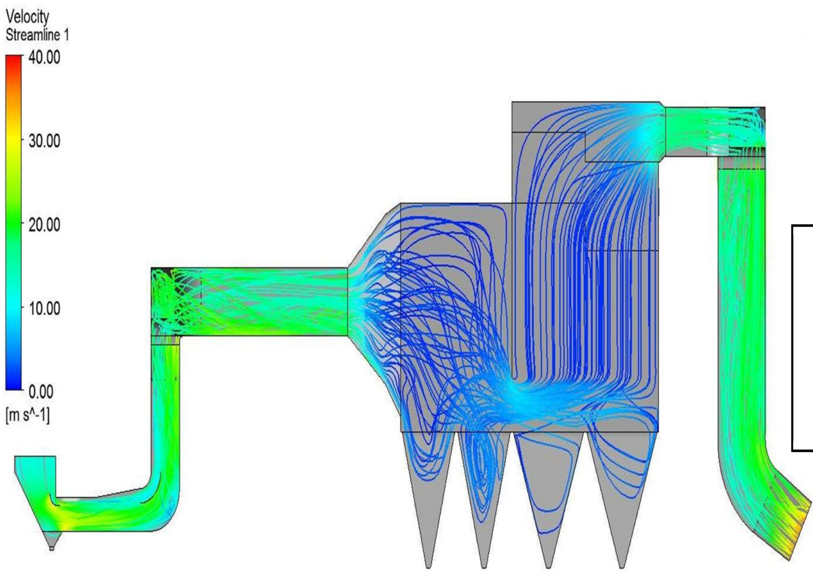


- Marked portion Shows the Flow separations at AH outlet & ESP Inlet Duct due to Sharp corner which results high turbulent flow

Figure illustrates velocity streamlines for Existing duct

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VIII Aug 2021- Available at www.ijraset.com

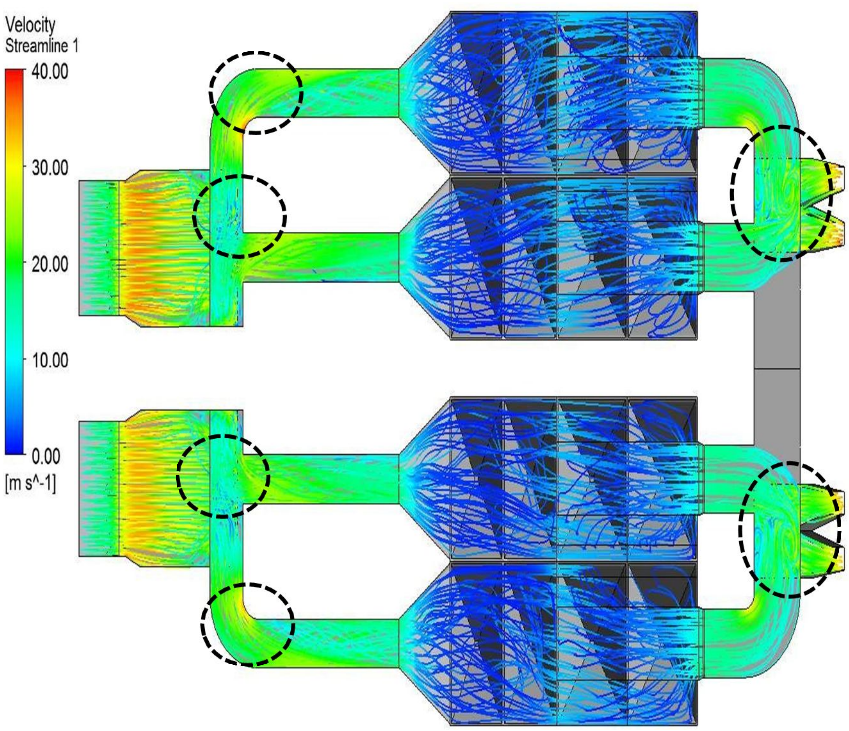
I. CFD Results Modified Duct



- After Modification in the Duct, flow separation & recirculation zones has been completely avoided which helps to reduce the pressure drop in Duct

Figure illustrates velocity streamlines for Modified Duct

J. CFD Results Existing Duct



- Marked portion Shows the Flow separations in Duct ESP Inlet Duct
- Due to Flow separation & high recirculation in ESP inlet duct creates High ΔP & Ash Accumulation
- These problems can be optimized be Re-design of Guide plates & Duct cross-section at few locations

Figure illustrates velocity streamlines for Existing Duct

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VIII Aug 2021- Available at www.ijraset.com

K. CFD Results Modified Duct

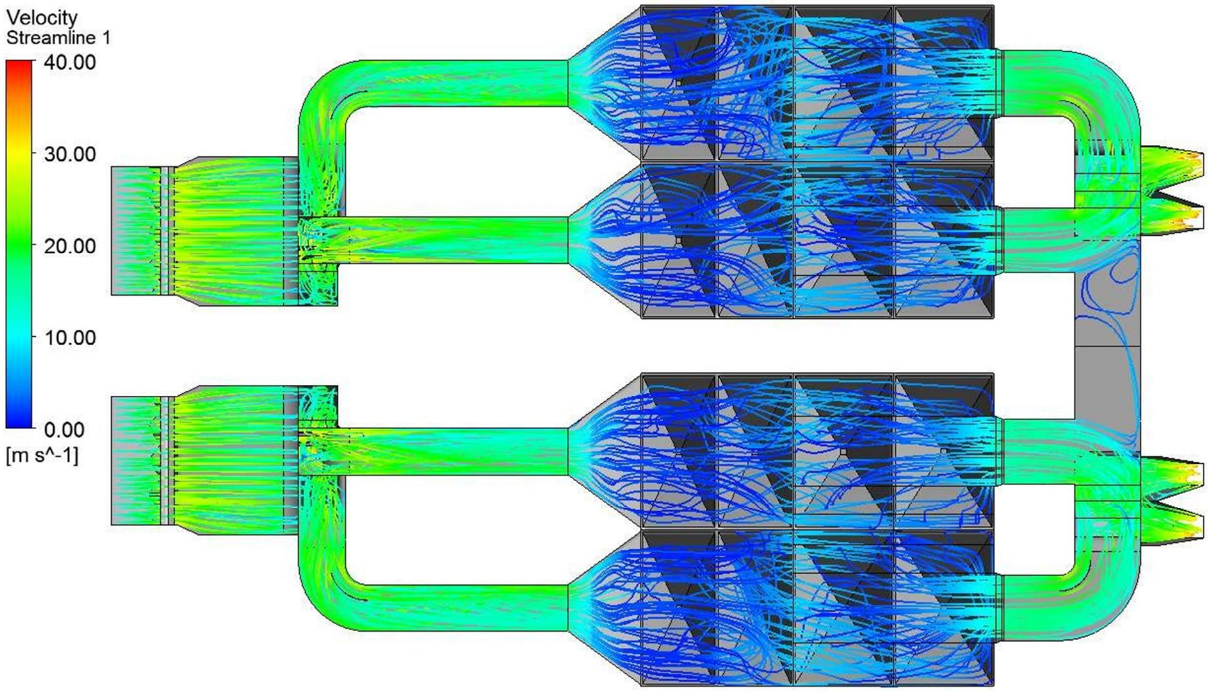
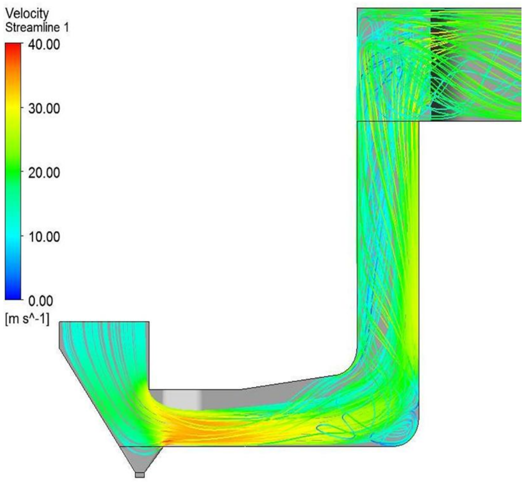


Figure illustrates velocity streamlines for Modified Duct

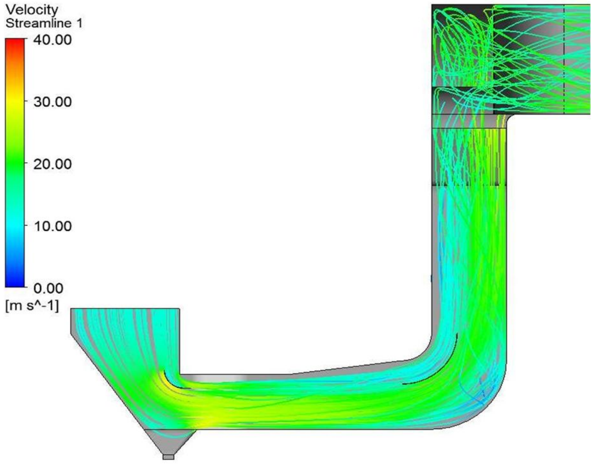
After Modification in the ESP Inlet Duct, flow separation & recirculation zones has been completely avoided .

L. CFD Results Comparision



Existing Duct -Streamline Plot

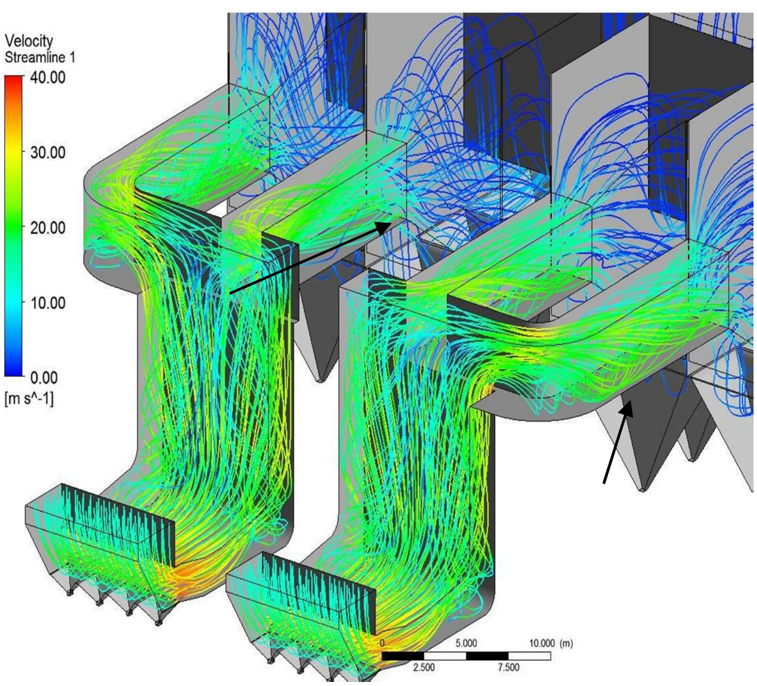
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VIII Aug 2021- Available at www.ijraset.com



Modified Duct -Streamline Plot

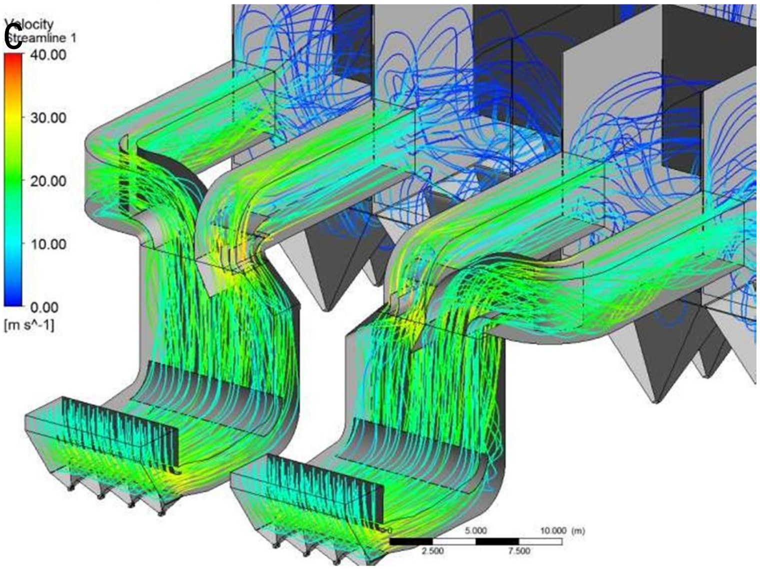
After Modification at APH outlet Duct,flow separation ,recirculation & Flow concentration zones has been completely optimized which helps to reduce the pressure drop & Ash accumulation in the duct

M. CFD Results Comparison



Existing Duct -Streamline Plot

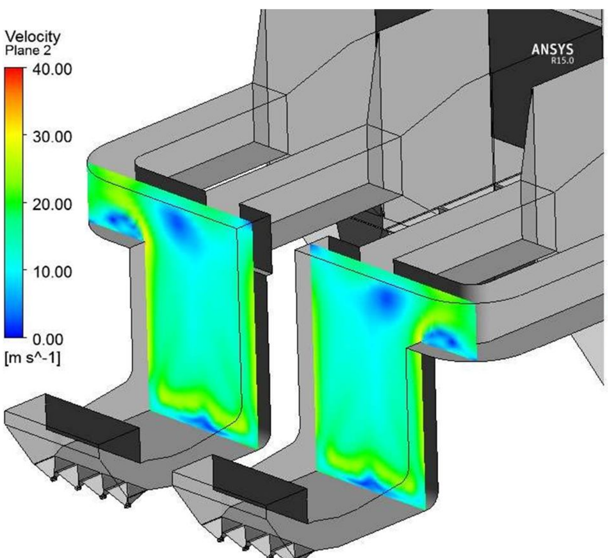
In Existing ESP Inlet Common duct, flow hits the Top surface of the duct & gets distributed in each pass as well as flow separation ,recirculation due to sharp corner, which has been completely optimized which helps to reduce the pressure drop & reduce Ash accumulation in the duct .It helps a lot to improve the Ash Flow distribution.



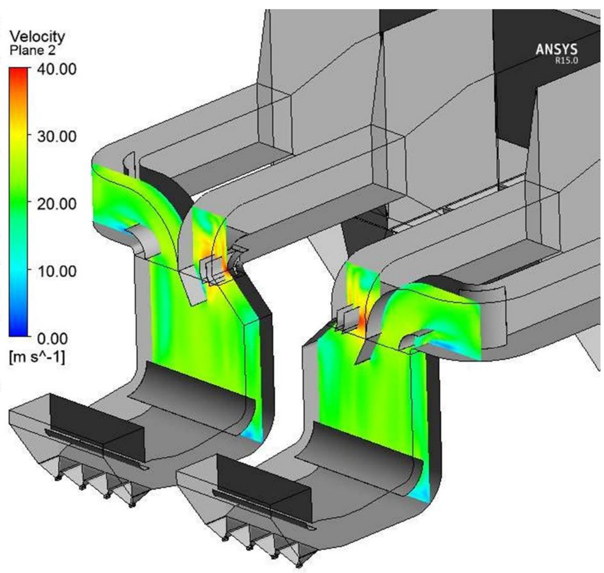
In Existing ESP Inlet Common duct, flow hits the Top surface of the duct & gets distributed in each pass as well as flow separation ,recirculation due to sharp corner, which has been completely optimized which helps to reduce the pressure drop & reduce Ash accumulation in the duct .It helps a lot to improve the Ash Flow distribution.

Modified Duct -Streamline Plot

N. CFD Results Comparision



Existing Duct -Velocity Plot

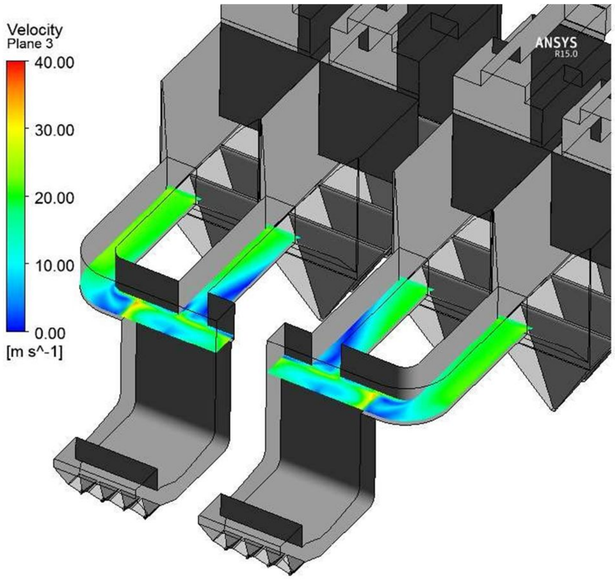


Modified Duct -Velocity Plot

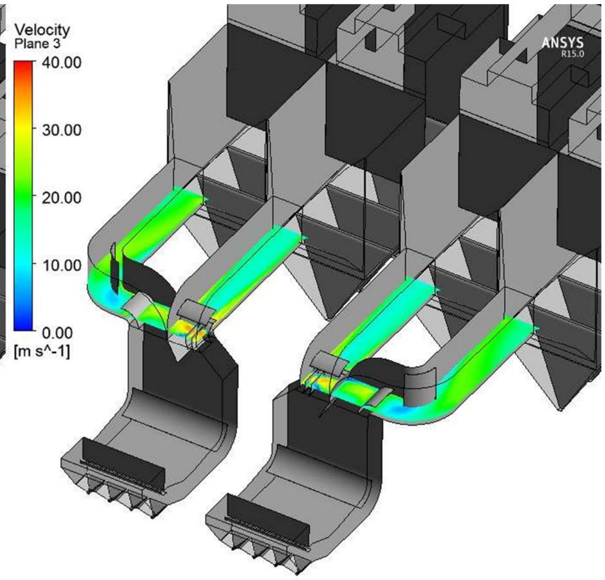
After Modification at ESP inlet Duct ,flow separation ,recirculation & Flow concentration zones has been minimized which
helps to reduce the pressure drop & Ash accumulation in the duct

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VIII Aug 2021- Available at www.ijraset.com

O. CFD Results Comparision



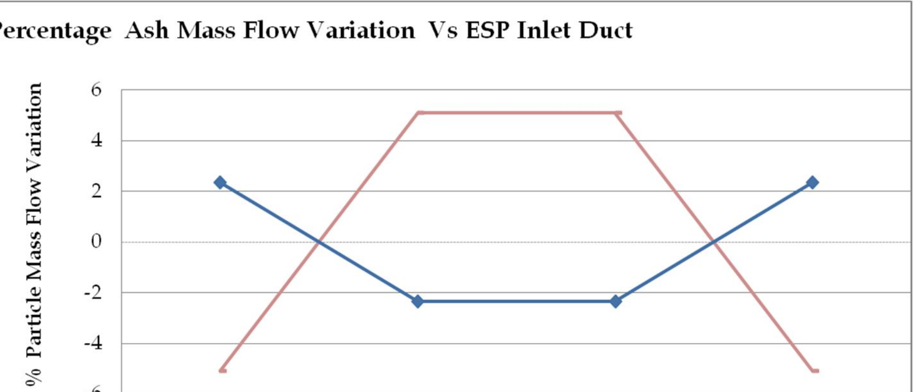
Existing Duct -Velocity Plot



Modified Duct -Velocity Plot

In Existing ESP Inlet duct, due to sharp corner of the ducting & absence of Guide plates , velocity is concentrated in particular area of the duct
After Modification ,the Flow has been improved which helps to balance the Ash as well as reduction in pressure drop .

III. ASH DISTRIBUTION RESULTS



	ESP INLET A PASS	ESP INLET B PASS	ESP INLET C PASS	ESP INLET D PASS
Existing % Ash Flow wrt Avg Ash Flow	-5.10	5.10	5.10	-5.10
Mod % Ash Flow wrt Avg Ash Flow	2.35	-2.35	-2.35	2.35

It can be observed that in Existing duct ,Ash is higher by 5.10% in Duct B & C pass wrt Avg Ash Flow rate
After modification, Ash flow has been reduced to 2.35% in Duct B & C pass wrt Avg Ash Flow rate .

IV. CONCLUSION

A. From Existing Duct CFD Analysis it can be conclude that there is high flow separation , recirculation & dead zones in the duct which creates the high pressure drop & ash accumulation & mass imbalance problem. This problems can be optimized by design of proper Guide vanes & Duct cross-sections.

B. From Existing Duct Ash particle CFD analysis , it can be observed that ,there are Ash accumulation zone in the duct due the drop down of velocity & recirculation , ESP inlet B & C Pass having Higher particle flow rate.

C. By Designing the optimized Guide Vane and Duct Plates using CFD ,the flow is nearly uniform with optimum turbulence & completely avoiding recirculation zones, which results in reduction in pressure drop.

D. Due to flow improvements in the duct ,ash accumulation has been totally minimized .

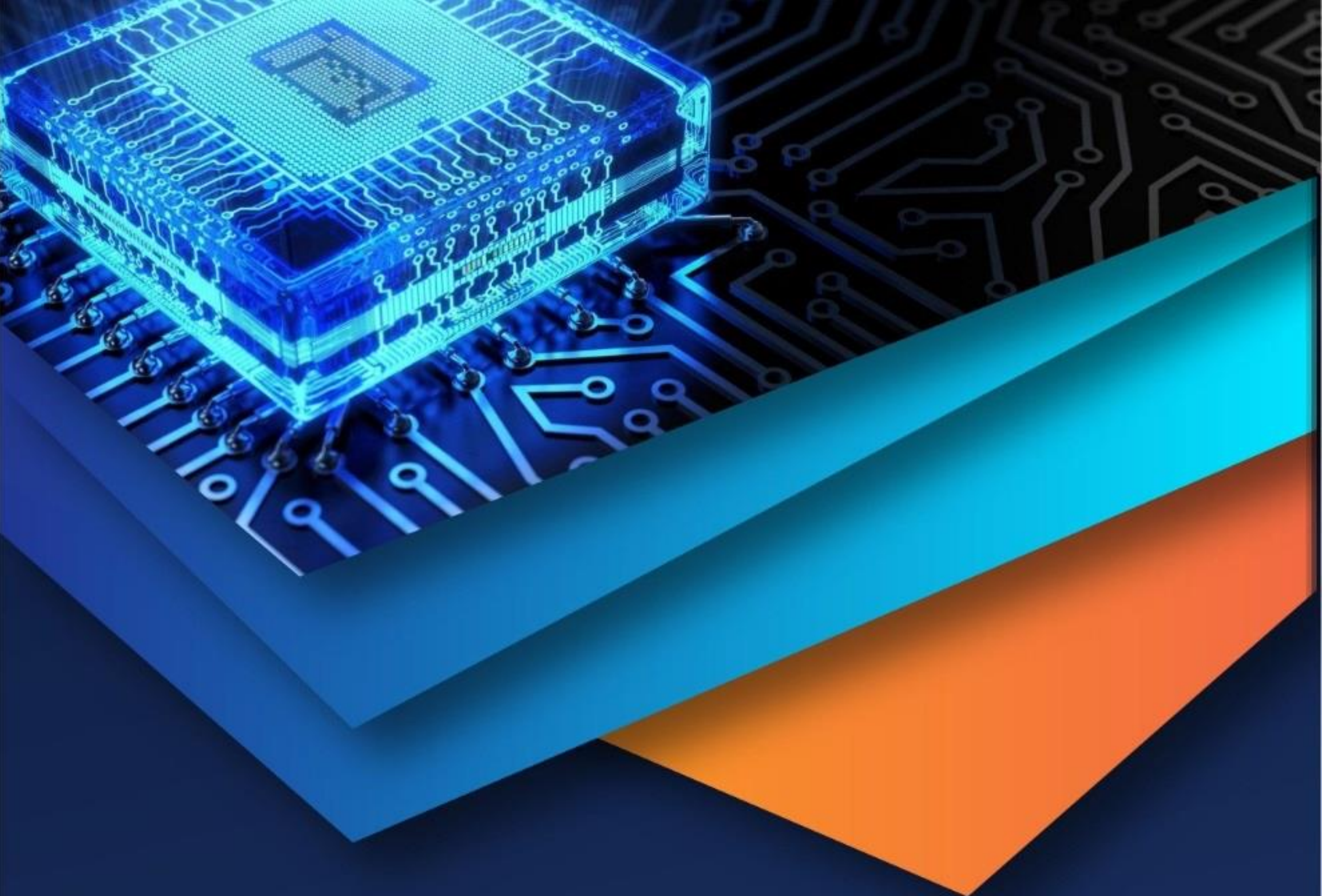
E. After modification, Gas flow in all pass of the duct will be balanced within 5 % of the avg.flow rate & ash particle flow will be balanced within 5%.

F. Pressure drop reduction from AH outlet to ID Fan inlet is 25 mmWC for the Applied flow rate.

G. Refractory is recommended for the high velocity zones to prolong the life of guide plates & duct plates.

REFERENCES

[1] https://www.researchgate.net/publication/283133237
[2] Power Plant Engineering by P K Nag.
[3] Brian J. Durmont, G. Robert, and P. E. Murdy, “CFD modeling of electrostatic precipitator”, Proceedings of 8th International Conference on Electrostatic Precipitation, Birmingham, Albama, USA, 2001
[4] C. Bhasker, “Flow simulation in electrostatic precipitator (ESP) ducts with turning vanes”, Advances in Engineering Software, vol. 42, Issue 7, pp. 501-512, 2011
[5] J. A. LaRose, and Allen Averill, “precipitator inlet particulate distribution flow analysis, industrial and environmental applications of fluid mechanics”, ASME FED vol. 186, pp. 91 – 99, 1994.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)