Improvement of ESP Efficiency using Computational Fluid Dynamics (CFD) BHEL - Perspective

REQUIREMENT FOR CFD

Rapid Innovation for the new Products – **To market quickly**

To make a better decision for an existing products – **To achieve profitability with quality**

Better Insight into the product behavior during development cycle – **To address top pressure like Time, Quality & Cost**

BENEFITS OF CFD

CFD reduces the design cycle and cost – Leads to Design improvements and increase in efficiency & Performance of the product.

CFD used to evaluate many different configurations and compare the output of the simulation.

Cost reduction by eliminating the requirement for many physical prototypes.

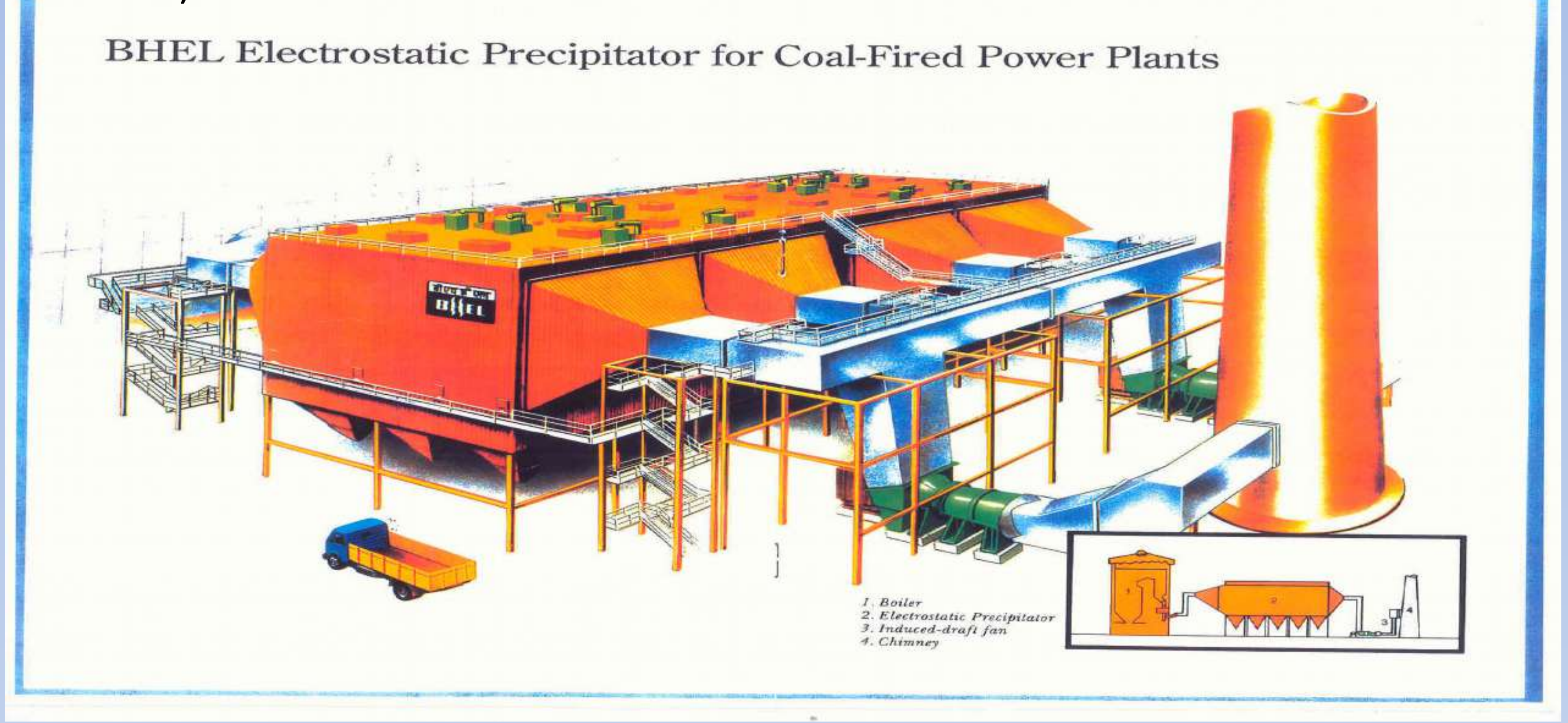
Many scenarios can be tested, with possible simulation at all conditions. (What-if-Scenarios)

Identifying root cause analysis leads to shorter trouble shooting

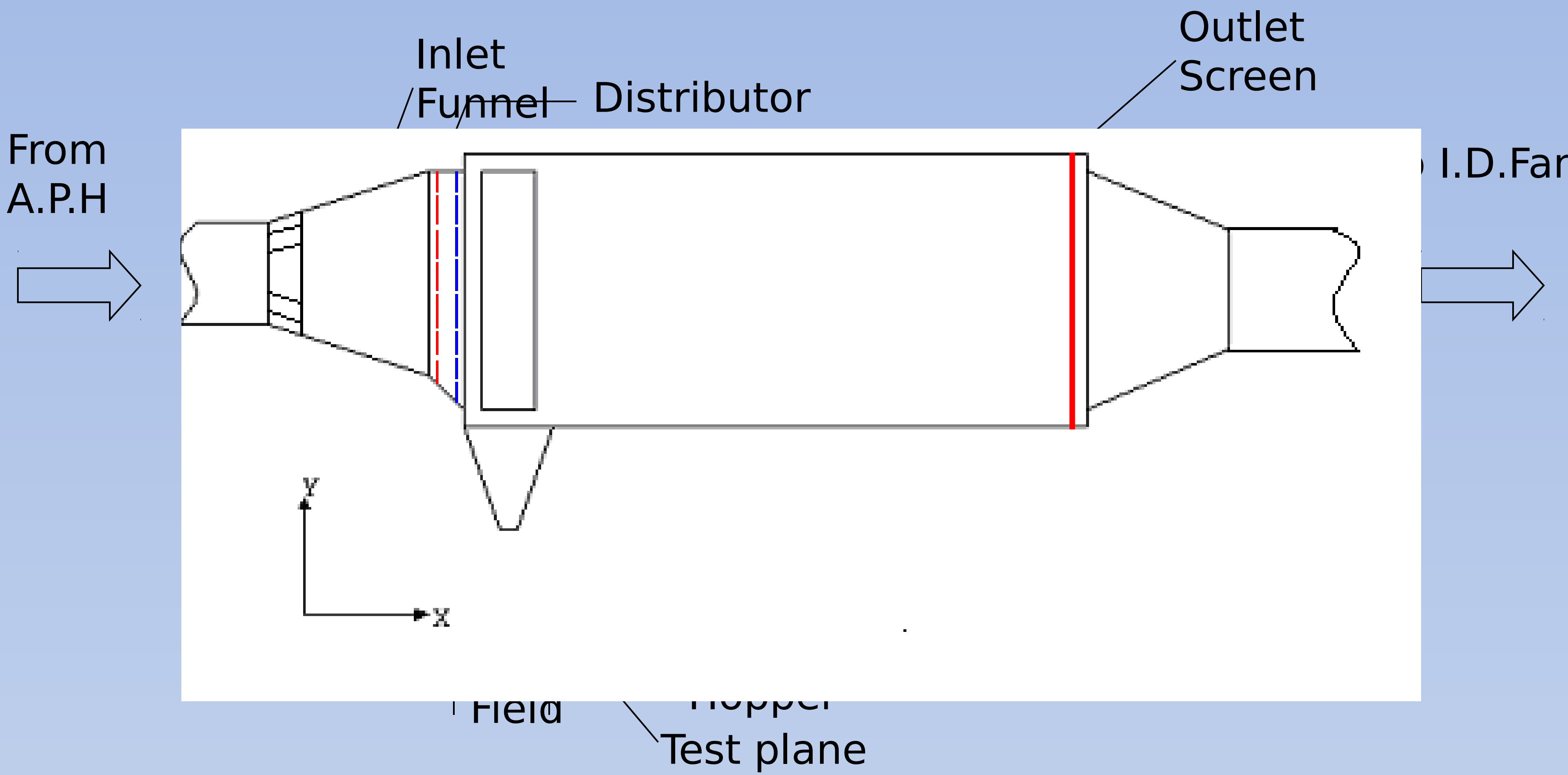
Electro Static Precipitator (ESP) is a pollution control equipment which removes the suspended particles present in the gas.

Dust collection is mainly dependent on the Flue gas distribution among and inside the **ESP**.

To achieve uniform gas distribution, flow control devices like GD Screen(s) plays a significant role in addition to splitter vanes, baffles etc.



Schematic Layout of ESP



Objective of ESP CFD modeling:

- To ensure equal flow & ash distribution among ESP passes
- To ensure uniform flow distribution inside the ESP
- To optimize the pressure drop across the ESP & ducting system.

To determine optimum location of guide vanes in Inlet ducts and deflector plates on ESP GD screens to achieve the above objectives

STANDARDS FOR CFD FLOW MODEL STUDY

- **Institute of Clean Air Companies(ICAC-EP-7)**

  within the treatment zone, near the inlet and outlet faces of the precipitator collection chamber, the velocity pattern shall have a minimum of 85 % of the velocities not more than 1.15 times the average velocity and 99 % of the velocities not more than 1.40 times the average velocity.

- **In the Inlet Duct System**

  The individual chamber volumetric flow should be compared with total system volumetric flow to ensure that the flow in each chamber is within ± 10 % of its theoretical share

CFD (Computational Fluid Dynamics) is the need of the hour – used globally everywhere in all type of Industries.

Ability to simulate real conditions and any physical flow condition in a **short period of time** at a **reduced cost.**

Experimental Model	CFD Model
Data to be extracted at a limited number of locations in the system	**To examine a large number of locations in the region of interest and yields a comprehensive set of flow parameters for examination and improvement.**
Lead time and cost involved are higher	**Substantial reduction of lead times and costs of new designs**
Controlled experiments are difficult to perform	**Ability to study systems where controlled experiments are difficult or impossible to perform and under hazardous condition**

CFD animations can also present characteristics that are difficult to quantify in a physical model (i.e., a visual tracking of injected ash particles through a duct).

CFD Analysis (For Inlet Duct system)

The CFD analysis of the flow inside ESP duct work is carried out using CFD software's to determine the optimum position of Guide plates/Guide vanes in order to meet the ICAC standard.

Experimental approach:

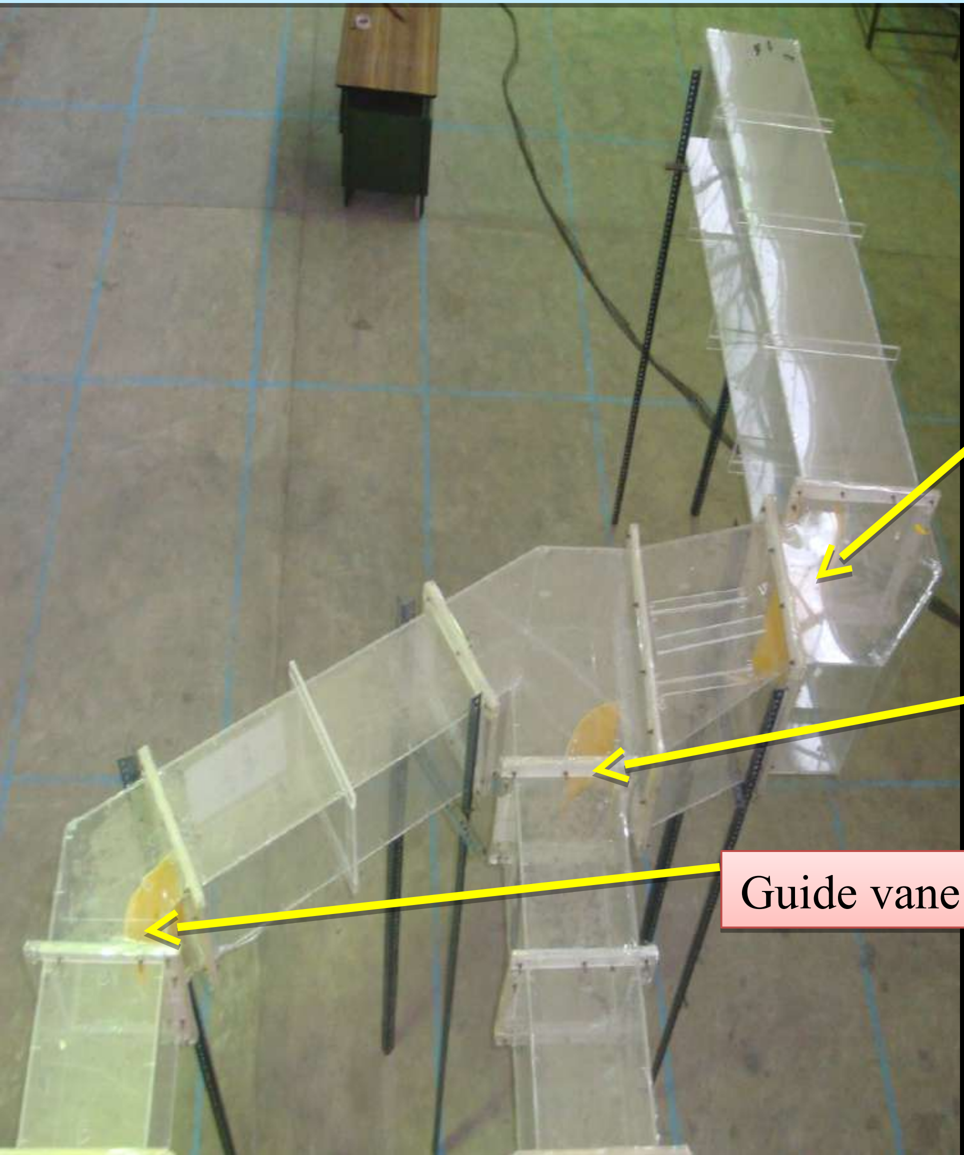
The location of Guide plates arrived by CFD is prescribed for the physical flow model set up thus eliminating the trial and error mechanism.

The measured experimental results can be validated with the help of CFD results.

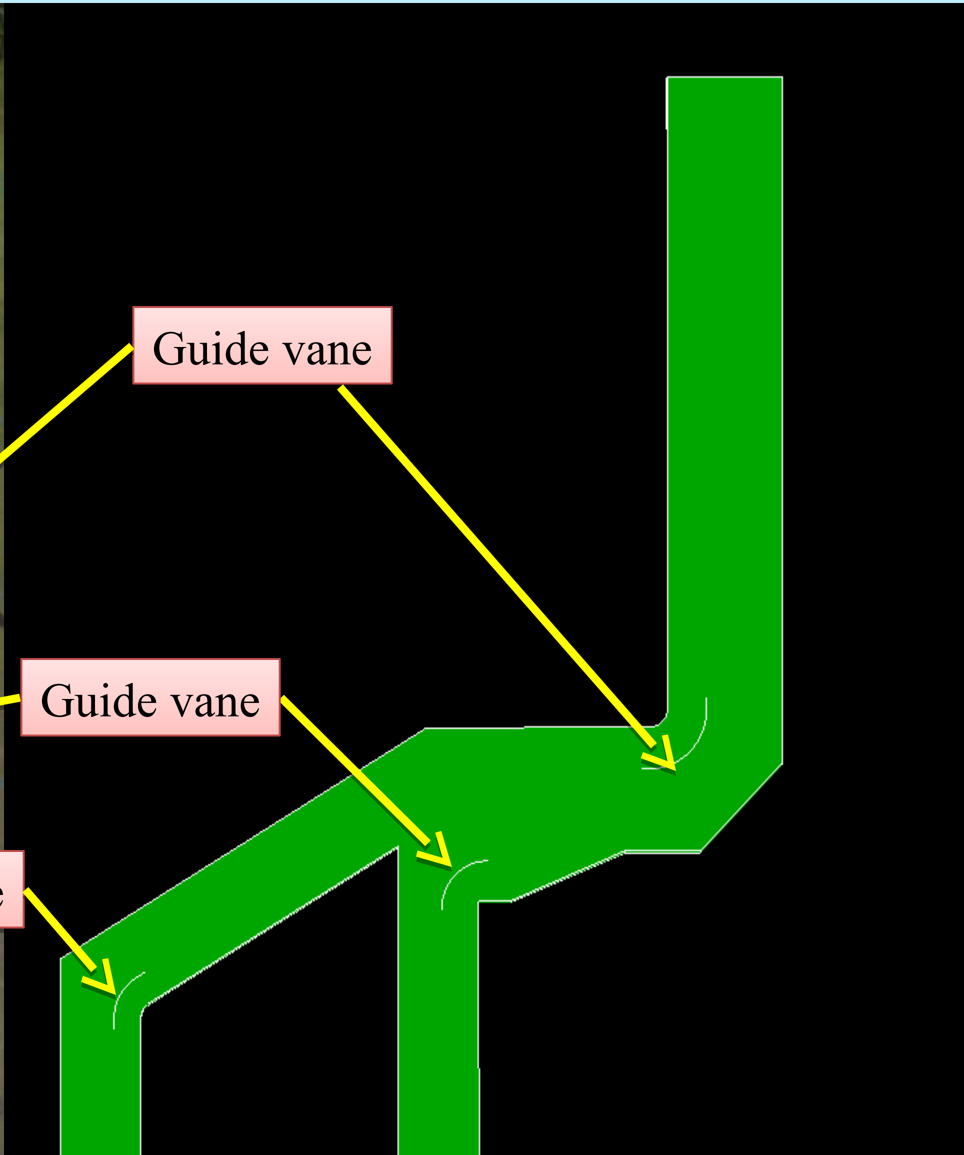
Typical physical scale model of an ESP at AQCS/BHEL, Ranipet

Geometry of ESP inlet with Guide vane

Experimental Model

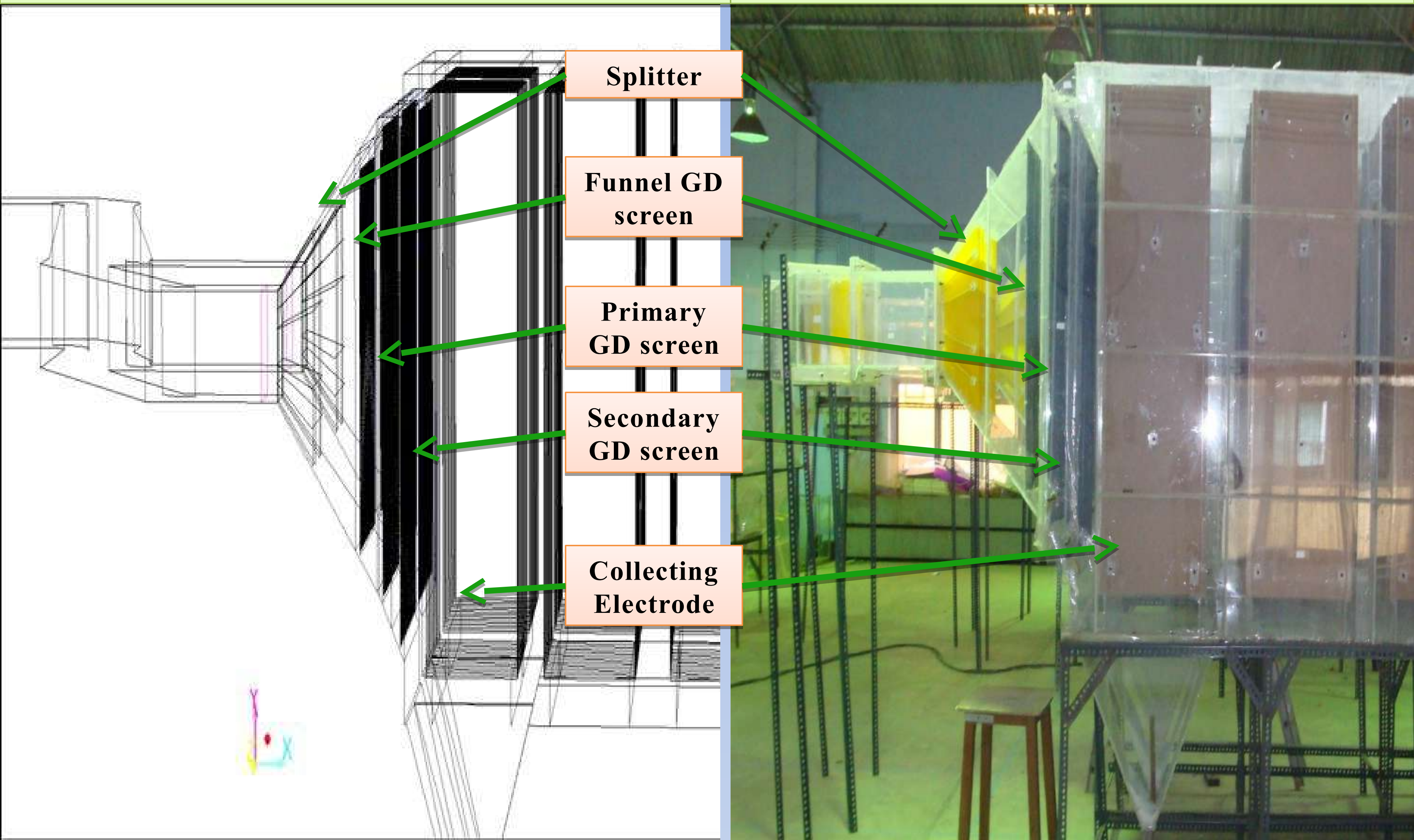


CFD Model



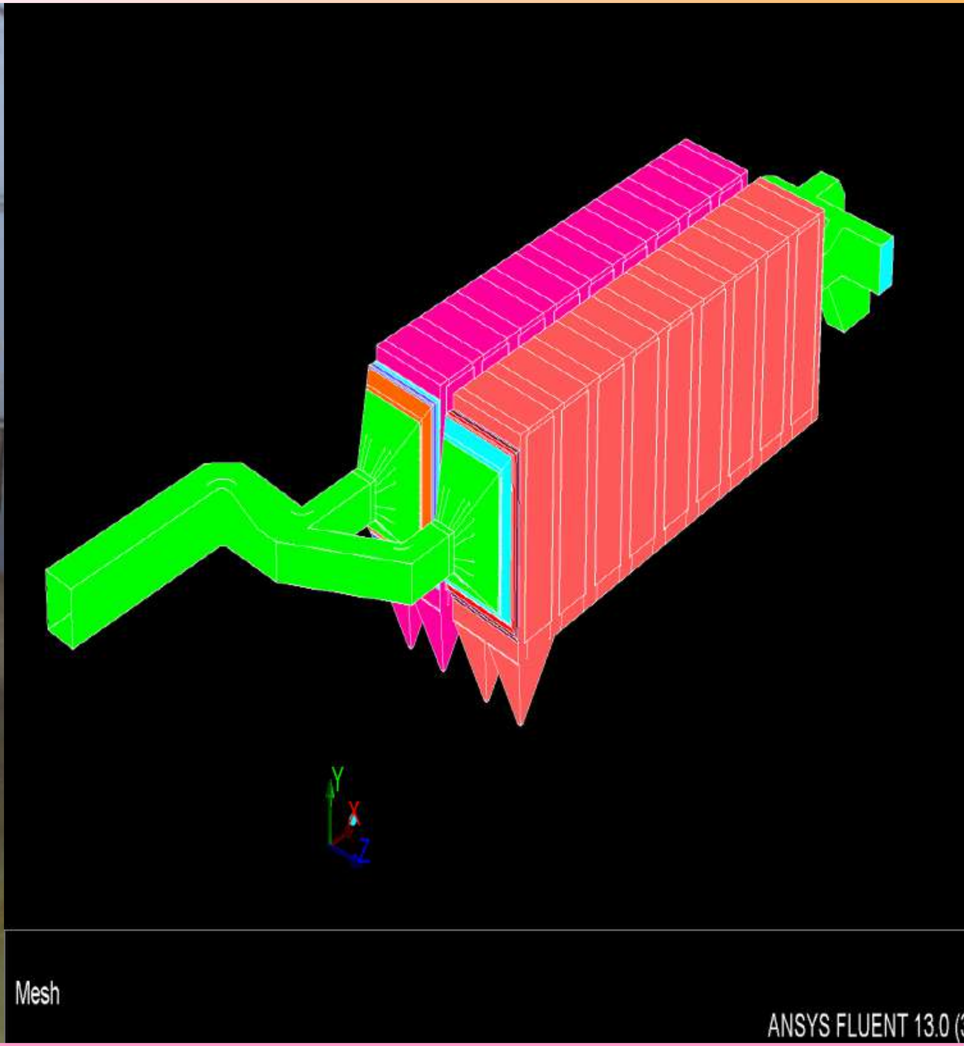
Geometry

CFD Model | Experimental Model

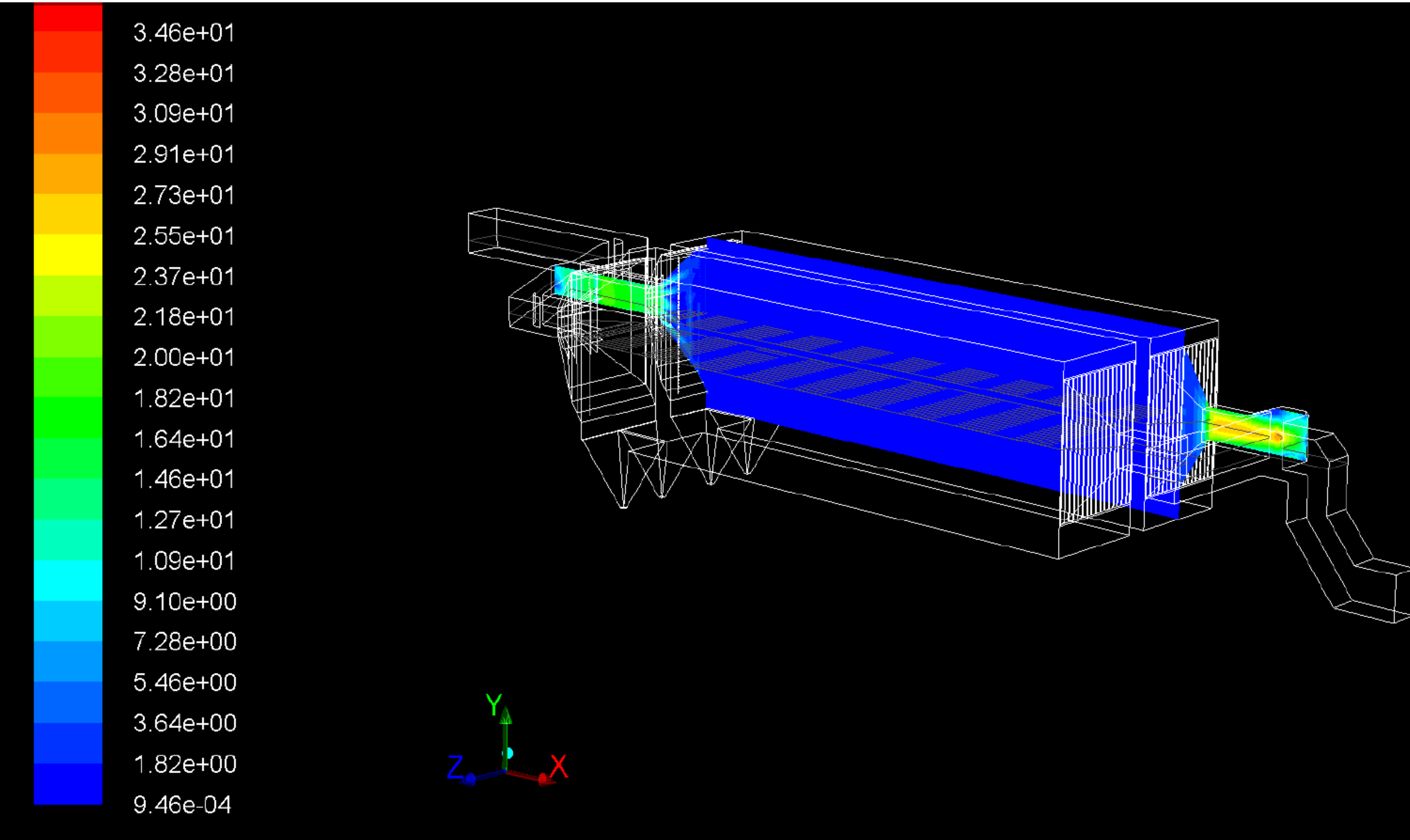


Geometry of ESP

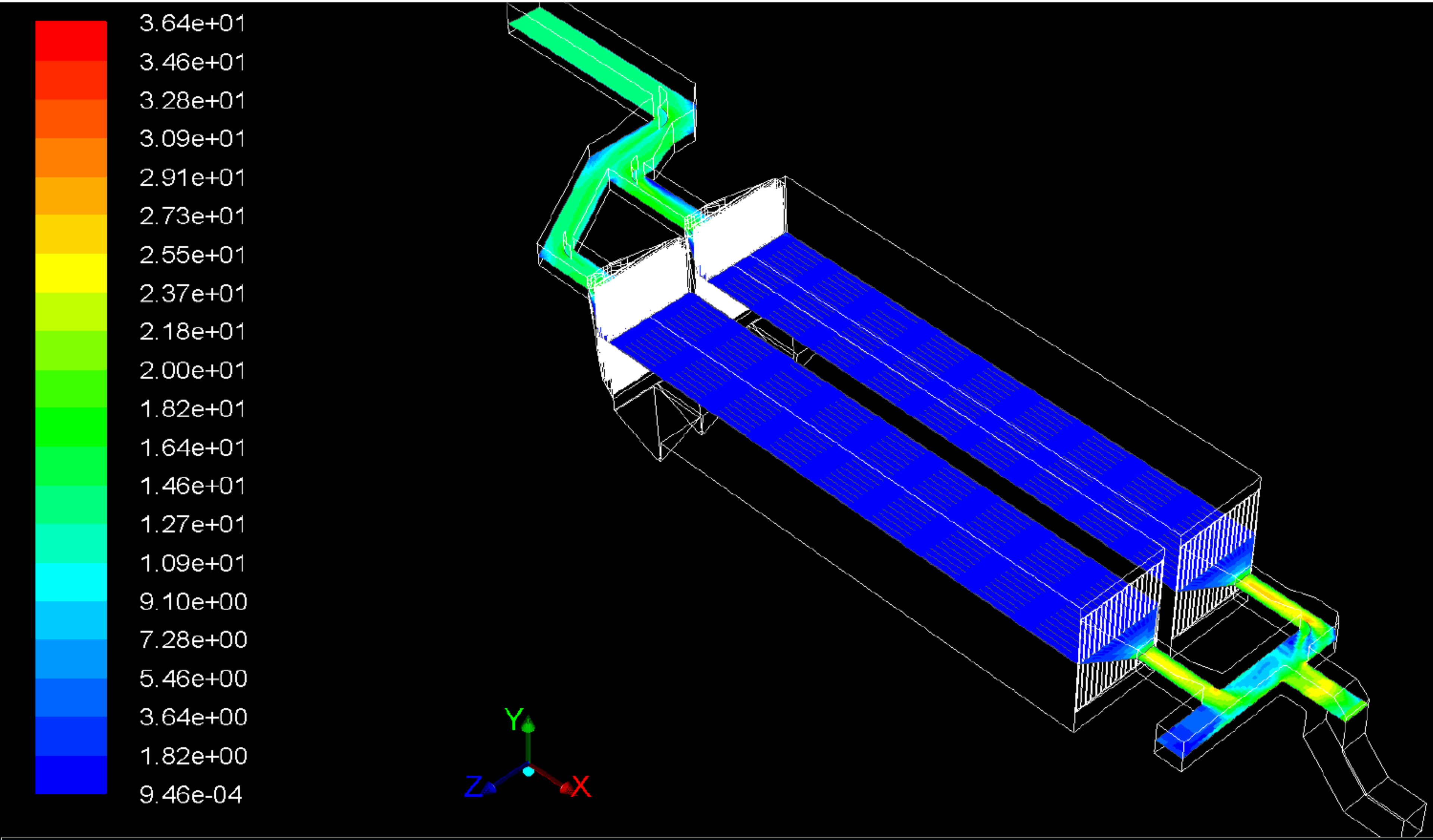
Experimental Model | **CFD Model**



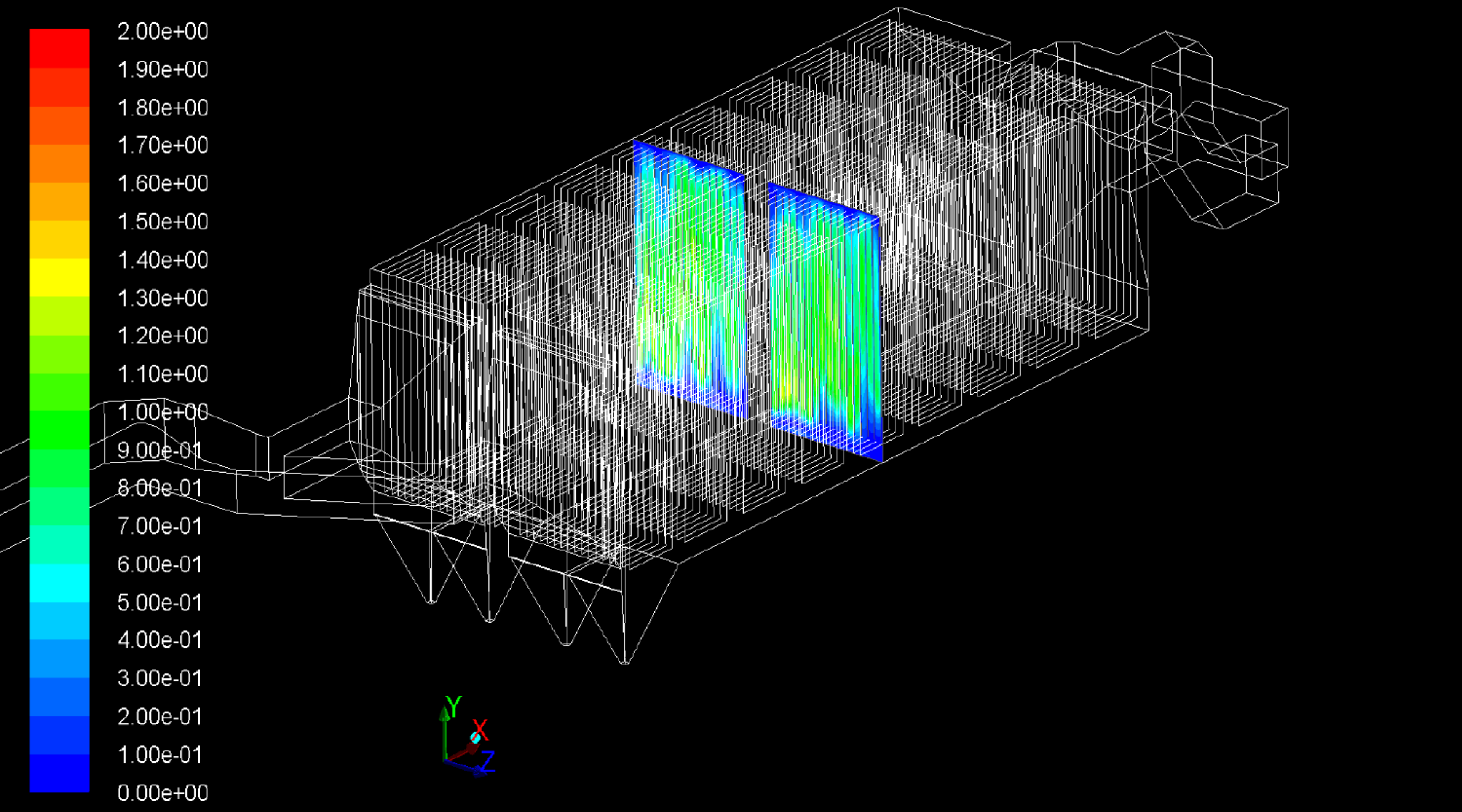
Velocity Contour at Vertical plane



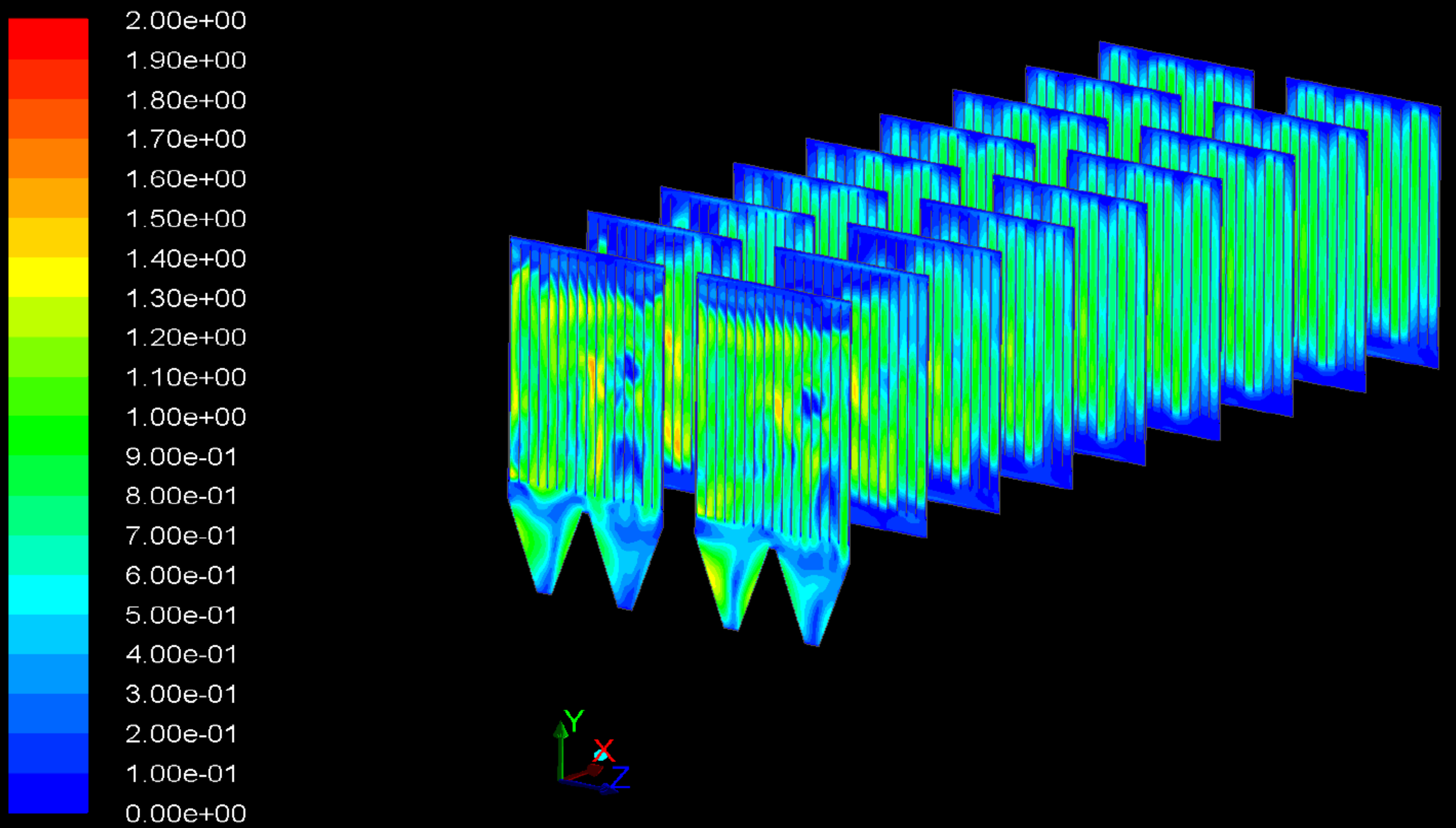
Velocity Contour at Horizontal Plane



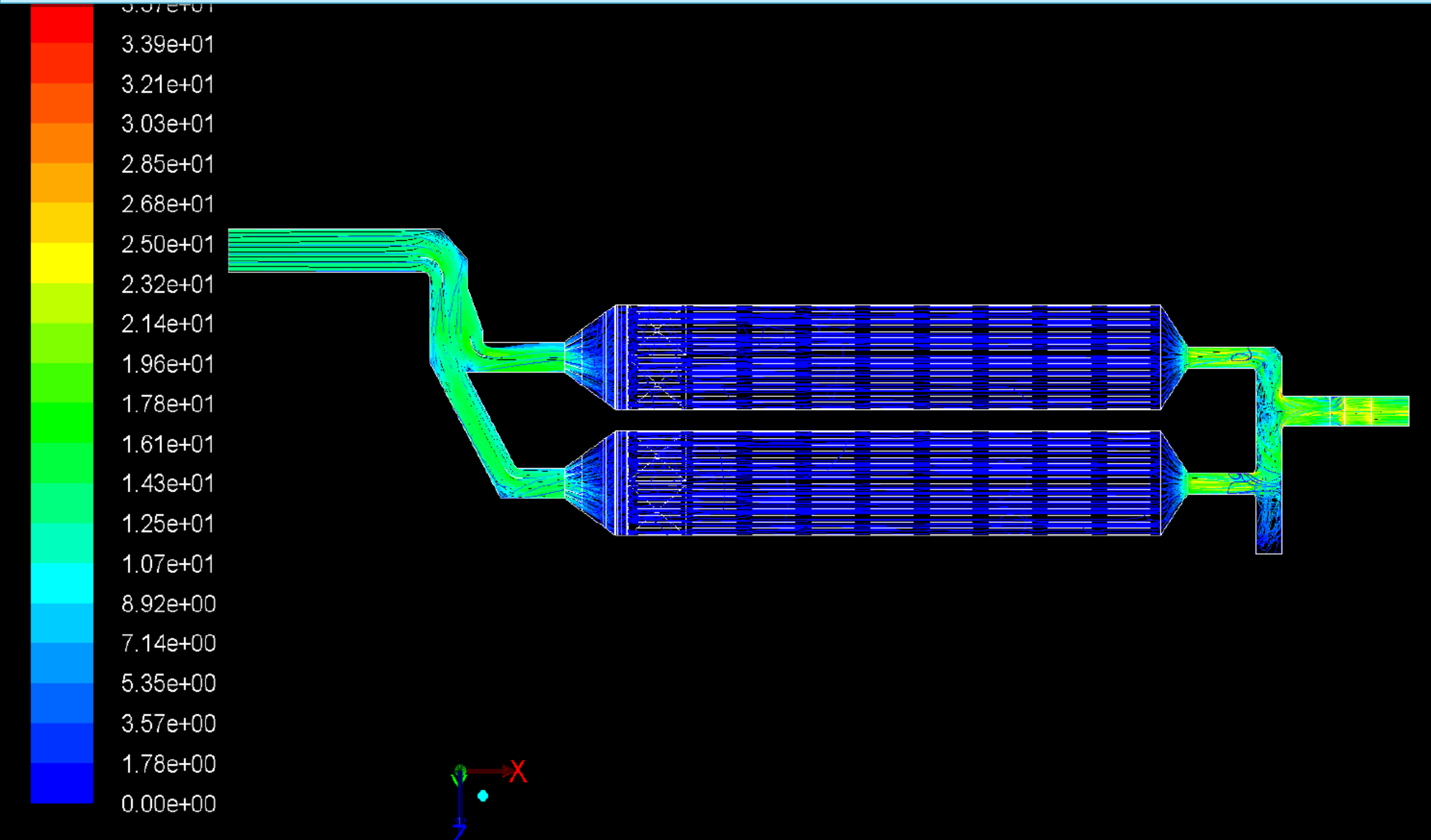
Velocity Contour at 5th Field – Isometric View



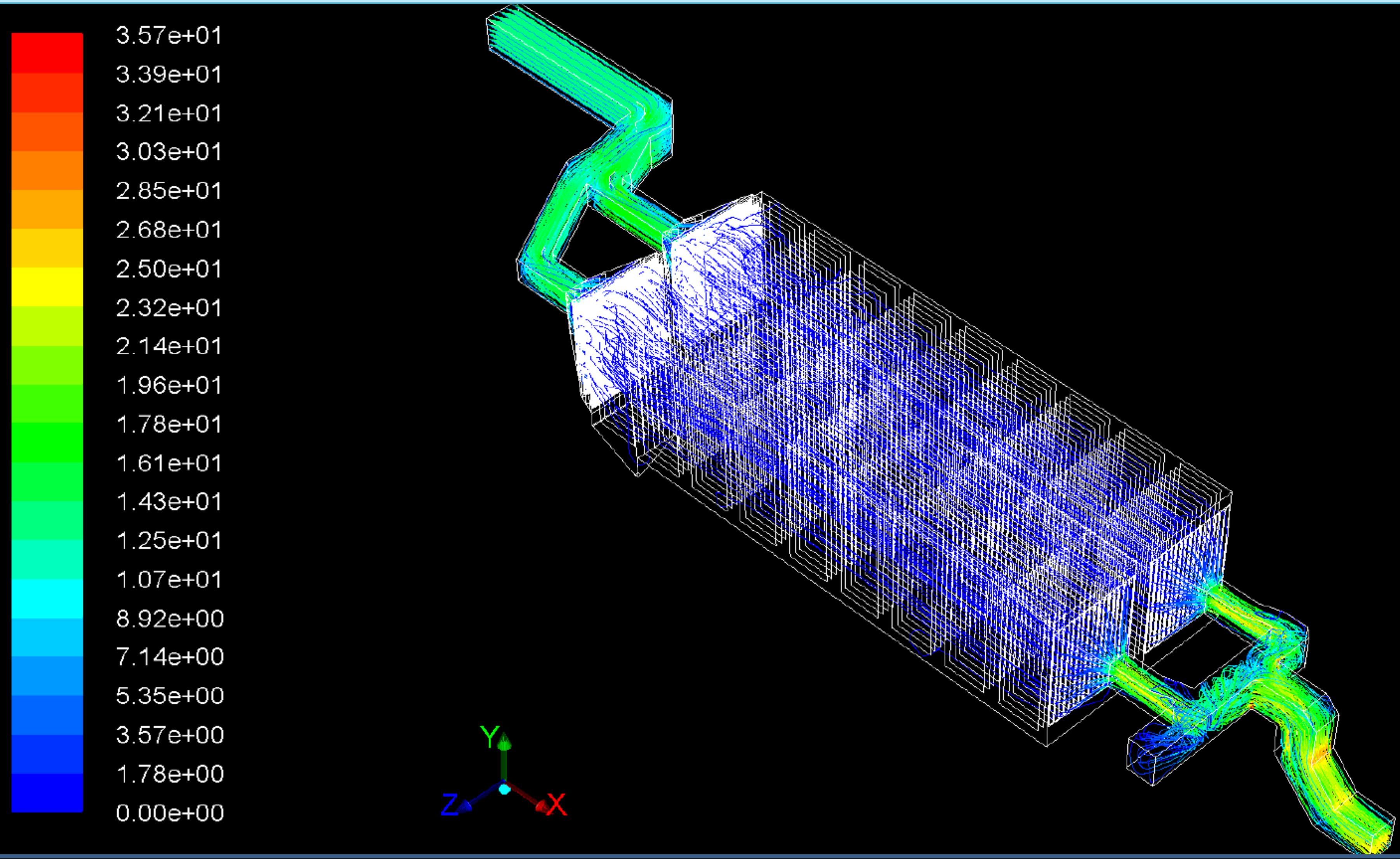
Velocity Contour in the ESP Fields - Isometric View



Velocity Path lines - Plane View



Velocity Path lines - Isometric View



RETROFITTING ESPs ………. CFD MODELING AS A TOOL

- **Possibility to provide validated designs right at the proposal stage.**
- **Flow pattern of existing products can also be analyzed to solve problems faced during operation.**
- **Results of CFD analysis reveal significant correlation to actual conditions.**
- **CFD is becoming an acceptable tool for design and problem solving.**

DIFFICULTIES FACED IN ESP RETROFIT

- **Restricted space availability to install new state of art ESP.**

- **Non-availability of required boiler shut down time to dismantle old dust collectors and to install new ESP.**

- **Congestion / complication of layout for routing flue gas ducts at inlet and outlet of ESP.**

- **Flow distribution requirement meeting Inter national standards.**

- **To meet MOEF Norms**

SOLUTION'S ?

- Additional collection area by adding new ESP in parallel to existing ESP.
- New ESP was installed in the space available adjacent to existing ESP.
- This approach was used to avoid longer shut down time of boiler.
- Flue gas flow was apportioned among new and original ESPs according to their available collection area.
- BHEL employ CFD techniques to propose sound designs to customers and also to study various site problems.

APPLICATION OF CFD ON ESP TO IMPROVE THE PERFORMANCE

mulation with Two Phase Flow

study the behavior of Ash particles)

ull Load and Part loading condition

study the Ash deposition in ID system)

se of Dampers at outlet ID system

control the flow distribution
for different Isolation cases)

nalysis of Pressure Drop

reduce loading of ID fan)