SCM ENGINEERING SERVICES

Technical Report on

CFD ANALYSIS OF AN ESP SYSTEM

Executed By:

S Sunil Krishna
sunilkrishna.s@scmengg.com

S Sampath Kumar
sampath.s@scmengg.com

Technical Director

Dr. Subhash Deo
subhash@scmengg.com

SCM Engineering Services
Plot #156, First Floor,
Motilal Nagar,
Begumpet,
Hyderabad 500019,
India.
Ph: 040 – 65449094
Email: projects@scmengg.com
URL: http://www.scmengg.com

CONTENTS

1. INTRODUCTION

The present study involves, CFD simulation of Electro Static Precipitator (ESP) used in thermal power stations to separate the carbon dust from the exhaust gas from the boiler.

Each Boiler has been connected with four ESPs through common inlet plenum and the outlets from ESPs are similarly connected to common outlet plenum.

Two fans take away the gas from the outlet plenum and send to the Chimney.

The CFD analysis helps in understanding the flow distribution within the duct and ESP and gives the insight look over positioning the guide vanes and flow distribution devices.

The study was carried out in the procedure given below,

1. CAD cleaning
2. Meshing
3. Fluid flow Analysis

2. SOFTWARE AND HARDWARE USED

2.1 Software

The analysis consists of Pre–processing, analysis and post processing followed by the result synthesis. Mesh generation being critical to the analysis was done using ICEM-HEXA. Boundary conditions definition and post processing results analysis were carried out using Star-CD.

2.2 Hardware

The hardware used for mesh generation, pre processing, post processing and the Star-CD solver was Intel based windows platform. The work was carried out on P4 processor, with 2 GB RAM.

3. GEOMETRY AND MESH GENERATION

3.1 CAD clean up and mesh generation

3.1.1 CAD Clean Up

In order to simplify and making the geometry ready for meshing, a CAD clean up procedure was done using CATIA and steps indicated below:

- Removing the unwanted surfaces and holes from the original igs file supplied.
- Surface trimming and using other techniques to get a closed surface
- Surface generation by closing edges

3.1.2 Mesh generation

Mesh was generated using ICEM - HEXA module of ICEM CFD software.

Boundary layers were made near the fluid wall-surfaces.

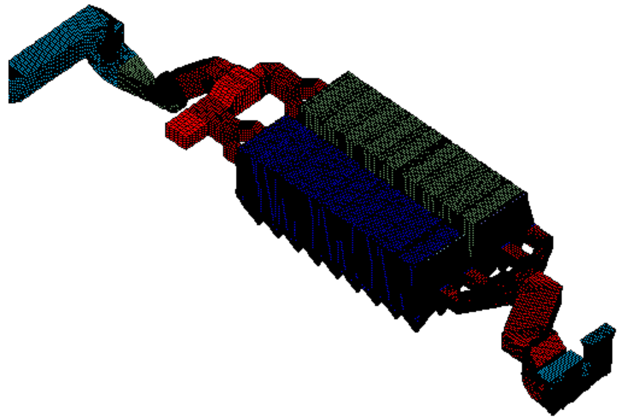


Fig 1. Mesh for fluid flow region

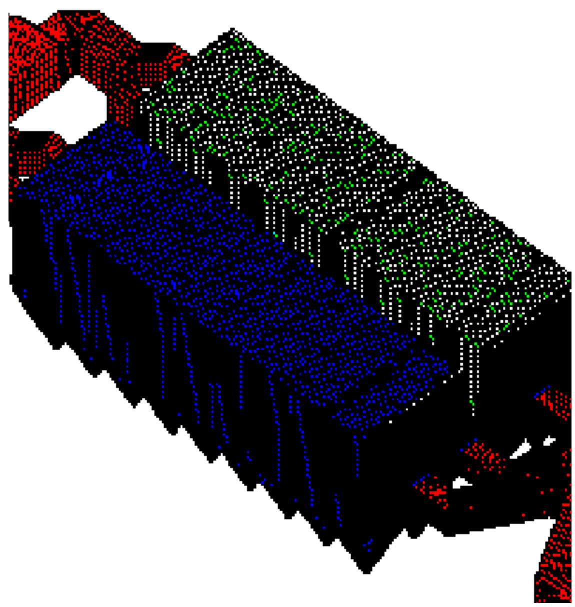


Fig 2. Closer view of the mesh

4. PROBLEM SET UP AND SOLVING

Spatial Discretisation Schemes used:

Upwind and MARS Differential Schemes.

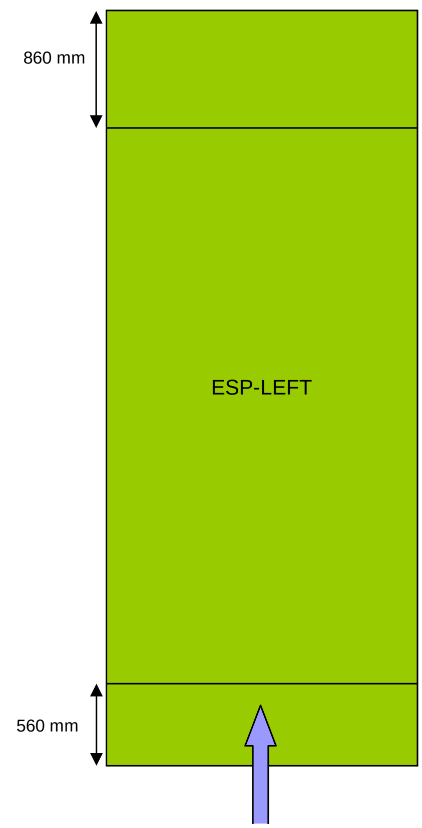
Convergence:

Mass, Momentum and Enthalpy convergence became to the order of 1.0e-4 after 3800 iterations.

5. RESULT OVERVIEW

As per CFD Model data		
Average	**S.D**	**C.V**
0.773356	0.063556	8.218162

As per Physical Model test data		
Average	**S.D**	**C.V**
1.016212	0.168559	16.587

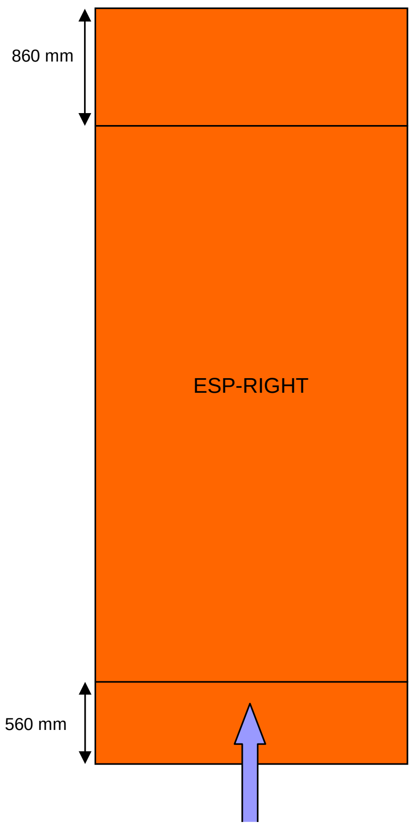


As per CFD Model data		
Average	**S.D**	**C.V**
0.763523	0.17144	22.45375

As per Physical Model test data		
Average	**S.D**	**C.V**
1.262183	0.222703	17.64424

As per CFD Model data		
Averag	**S.D**	**C.V**
0.78449	0.05669	7.2265

As per Physical Model test		
Averag	**S.D**	**C.V**
1.08314	0.16996	15.6916



As per CFD Model data		
Average	**S.D**	**C.V**
0.77863	0.16488	21.1766

As per Physical Model test data		
Average	**S.D**	**C.V**
1.13954	0.19182	16.8336

6. CONCLUSION

- The flow distribution and the pressure losses across the domains are computed for 100% rated flow condition.
- The gas phase velocities at 604 mm down stream and 1813 mm upstream within the ESP were obtained and the same were compared with the values obtained from the physical test.
- This comparison shows good agreement and the predicted results are inline with the physical test.
- The present simulation gives confidents that the current numerical model can be used for further numerical experiments