Shape optimization of Primary Air Duct of Thermal Power Plant Using CFD

N.J. Mohamad afshar farhan[1], M. Masi[1], P. Meenakshi sundaram[1], V.J. Mathan[1], S. Maniamramasamy[2]

1-UG Student, MAM School of Engineering, Trichy, Tamil nadu, India

2-Assistant Professor, MAM School of Engineering, Trichy, Tamil nadu, India

afsharfarhanjb001@gmail.com, masimurugesan341996@gmail.com, rksmaniyam@gmail.com

***Abstract*- Generally simple shape ducts are used in thermal power plants to ease of manufacturing however modification has to be done in order to accommodate interfacing of equipments which are associated with plant operation leading to higher pressure drop, higher power consumption and flow misdistribution zones having higher or lower velocity. To readdress this situation, baffles, guide vanes and other internals components are used to smoothen the flow through ducts especially in bend. The basic disadvantage in using baffles get punctured/eroded due to impact of high velocity ash particles in flue gas ducting and the effectiveness of baffles is lost in short duration. So using guide vanes and deflector plates will be more efficient in coal handling. We have optimized primary air duct to overcome the above disadvantages, the bend of the duct is filleted and guide vanes are attached in the middle of the duct bends in such a way that a more streamlined flow is maintained across any cross-section.**

Keywords- CFD, primary air duct, guide vanes & deflector plate.

I. INTRODUCTION

1.1. Power Plant

A power plant is a facility for the generation of bulk electric power. Power plants produce electric energy from another form of energy. The type of converted energy depends on the type of power plant. Each type of power plant poses a distinct set of advantages and drawbacks. The energy for power plants comes from several sources. Some sources are nonrenewable, such as natural gas, oil, coal and nuclear fuels, while others are renewable, such as manure, straw and wood. The cost or efficiency of generation depends on the power rating of the source, how long it is used and the price of an electrical unit. The most common fuels used in power plants are fossil fuels, nuclear fuels and renewable biomass (straw, manure and wood).

1.2. Thermal Power Plants

A thermal power station is a power plant in which heat energy is converted to electric power. In most of the places in the world the turbine is steam-driven. Water is heated, turns into steam and spins a steam turbine which drives an electrical generator. After it passes through the turbine, the steam is condensed in a condenser and recycled to where it was heated; this is known as a Rankine cycle. The greatest variation in the design of thermal power stations is due to the different heat sources, fossil fuel dominates here, although nuclear heat energy and solar heat energy are also used. Some prefer to use the term *energy center* because such facilities convert forms of heat energy into electrical energy.

1.3. Duct Systems in Power Plant

Process duct work conveys large volumes of hot, dusty air from processing equipment to mills, baghouses to other process equipment. Process duct work may be round or rectangular. Although round duct work costs more to fabricate than rectangular duct work, it requires fewer stiffeners and is favored in many applications over rectangular ductwork .The air in process duct work may be at ambient conditions or may operate at up to 900 °F (482 °C). Process ductwork varies in size from 2ft diameter to 20ft diameter or to perhaps 20ft by 40ft rectangular. Large process ductwork may fill with dust, depending on slope, to up to 30% of cross section, which can weigh 2 to 4 tons per linear foot. Round ductwork is subject to duct suction collapse, and requires stiffeners to minimize this but is more efficient on material than rectangular duct work. There are no comprehensive, design references for process duct work design. The ASCE reference for the design of power plant duct design gives some general guidance on duct design, but does not specifically give designers sufficient information to design process duct work.

II. COMPUTATIONAL FLUID DYNAMIC

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of

- mathematical modeling (partial differential equations)
- numerical methods (discretization and solution the choice of numerical algorithms and data structures
- linear algebra tools, stopping criteria for iterative solvers
- discretization parameters (mesh quality, mesh size,

time step)
- cost per time step and convergence rates for outer iterations
- programming language (most CFD codes are written in Fortran)
- many other things (hardware, vector9ization, parallelization etc.)
- The quality of simulation results depends on the mathematical model and underlying assumptions
- approximation type, stability of the numerical scheme mesh, time step, error indicators, stopping criteria.

2.1. Post processing and Analysis

Post processing of the simulation results is performed in order to extract the desired information from the computed flow field

- calculation of derived quantities (stream function, vorticity)
- calculation of integral parameters (lift, drag, total mass)
- visualization (representation of numbers as images)
- 1D data function values connected by straight lines
- 2D data streamlines, contour levels, color diagrams
- 3D data cut lines, cut planes, iso surfaces, iso volumes
- arrow plots, particle tracing, animations . . .
- Systematic data analysis by means of statistical tools
- Debugging, verification, and validation of the CFD model

III. BASE DUCT DESIGN

3.1. Duct Geometry

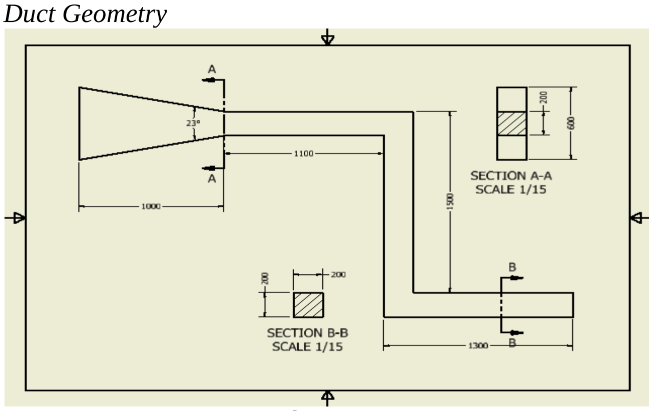

Fig.1. Base duct geometry

3.2. Boundary Condition

Velocity Inlet-2m/s
Velocity Outlet-2.2m/s
Pressure Inlet-609.513pa
Pressure Outlet-0pa

3.3. Result

A. Static Pressure Contour

The following figure shows the static pressure contours for various analyses with base duct configuration. The figure shows the variation between the minimum and maximum pressure values across the entire length of the base duct section taken into consideration.

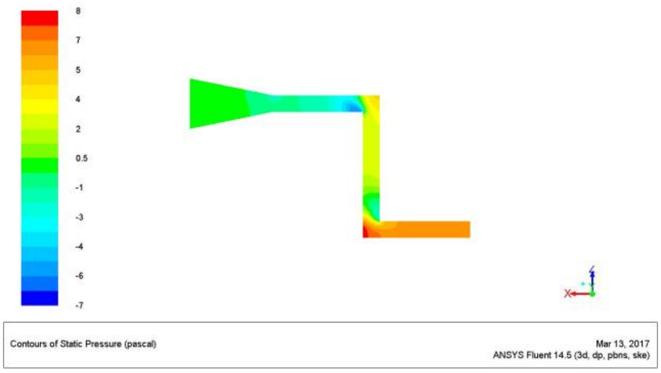

Fig.2. Static Pressure Contour

B. Velocity Contour

The following fig shows the velocity for various analyses with various duct configuration. The figure shows the variation between the maximum and minimum velocity values across the entire length of the duct section and around the enclosure surrounding the duct area taken into consideration. Also these contours show the velocity variation in the flow inside the duct.

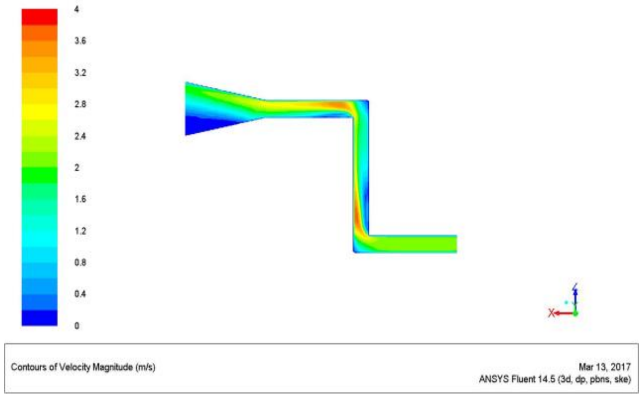

Fig.3. Velocity contour

IV. OPTIMIZATION 1

4.1. Duct Geometry

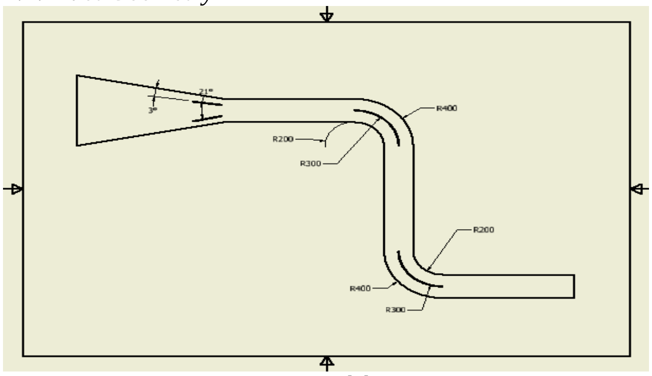

Fig.4. Optimized duct geometry

4.2. Boundary Condition

Velocity inlet-2m/s
Velocity outlet-1.83m/s
Pressure inlet-169.39pa
Pressure outlet-0pa

4.3. Result

A. Static Pressure Contour

The following figure shows the static pressure contours for various analyses with first optimized duct configuration. The figure shows the variation between the minimum and maximum pressure values across the entire length of the first optimized duct section taken into consideration

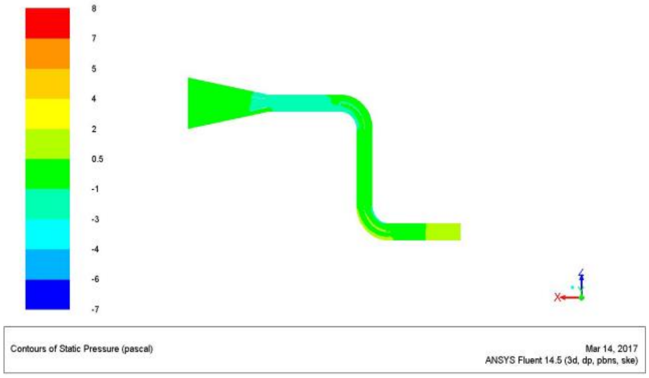

Fig.5. Static pressure contour

B. Velocity Contour

The following fig shows the velocity for various analyses with first optimized duct configuration. The figure shows the variation between the maximum and minimum velocity values across the entire length of the first optimized duct section and around the enclosure surrounding the duct area taken into consideration. Also these contours show the velocity variation in the flow inside the first optimized duct.

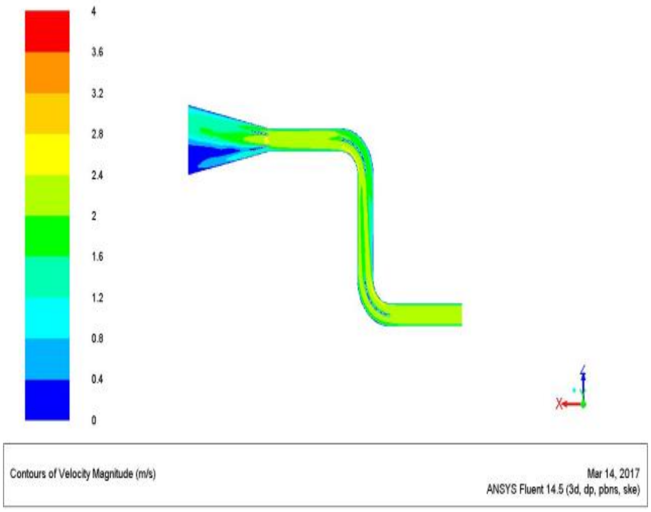

Fig.6. Velocity Contour

V. OPTIMIZATION 2

5.1. Duct Geometry

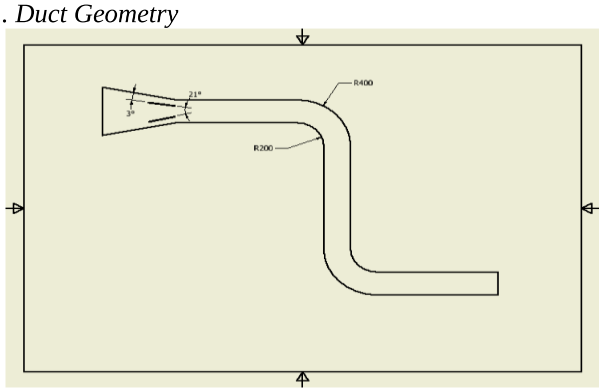

Fig.7. Optimized Duct Geometry 2

5.2. Boundary Condition

Velocity inlet-2m/s
Velocity outlet-1.5m/s
Pressure inlet-96.06pa
Pressure outlet-0pa

5.3. Result

A. Static Pressure Contour

The following figure shows the static pressure contours for various analyses with second optimized duct configuration. The figure shows the variation between the minimum and maximum pressure values across the entire length of the second optimized duct section taken into consideration.

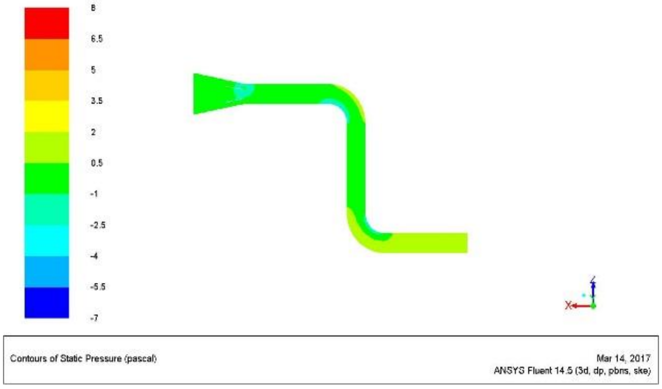

Fig.8. Static pressure contour

B. Velocity Contour

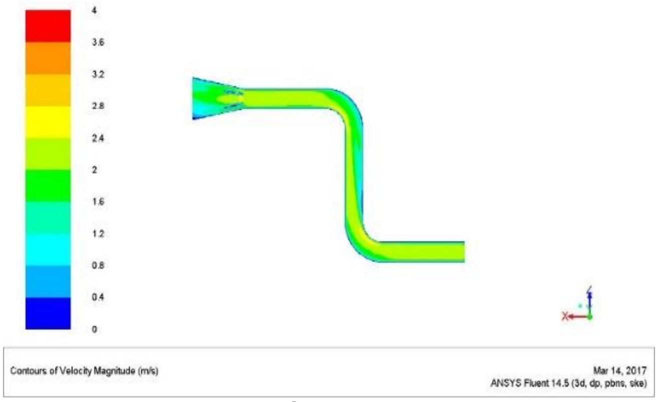

Fig.9. Velocity contour

Maximum pressure-96.06pa
Minimum pressure -0pa

TABLE I
Velocity And Pressure Results

	BASE	OPTIMIZATION 1	OPTIMIZATION 2
Iteration	1000	1000	1000
Velocity Inlet	2m/s	2m/s	2m/s
Velocity Outlet	2.2m/s	1.83m/s	1.5m/s
Inlet Area	2m	2m	2m
Pressure Inlet	609.513 pa	169.39pa	96.06pa
Pressure Outlet	0pa	0pa	0pa

VI. RESULT& SUMMARY

1. Base Duct Design

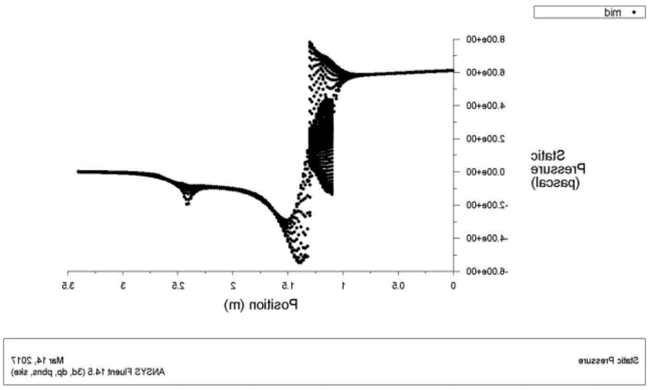

Fig .10. Static Pressure Contour

Maximum pressure -609.513pa
Minimum pressure -0pa

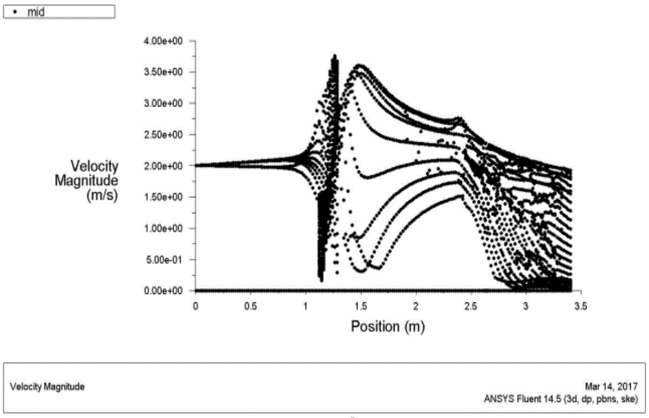

Fig.11. Base velocity contour

Maximum velocity -2.2m/s
Minimum velocity -2m/s

2. Optimized Design 1

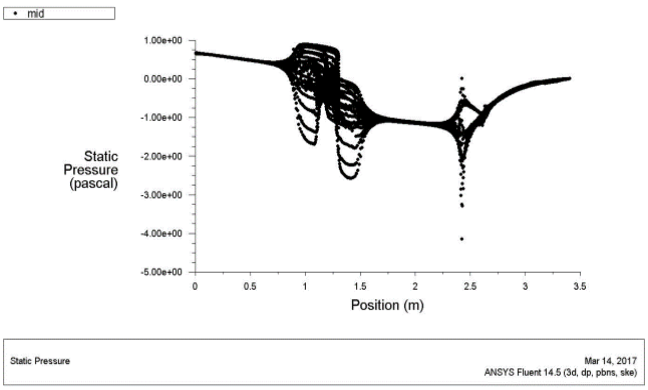

Fig .12. Static pressure contour

Maximum pressure -169.39pa
Minimum pressure-0pa

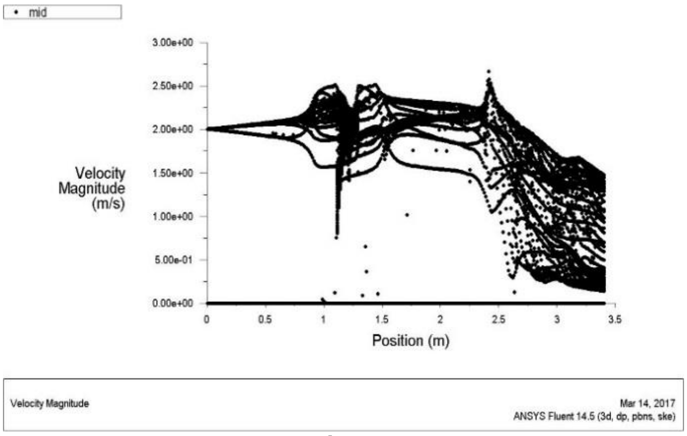

Fig.13. Velocity Contour

Maximum velocity -2m/s
Minimum velocity -1.83m/s

3. Optimized Design 2

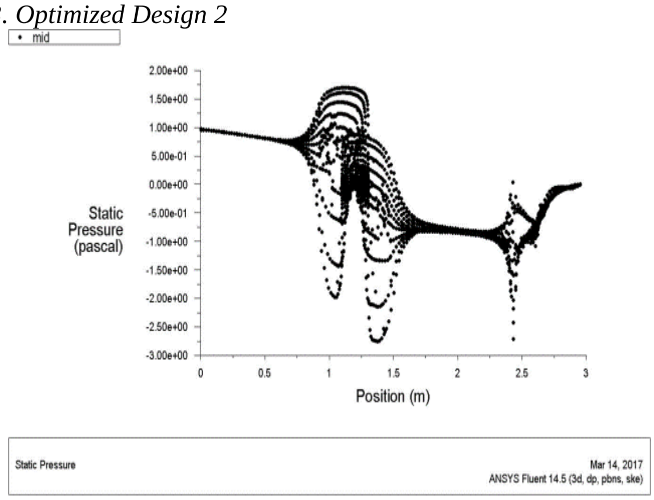

Fig.14. Static pressure contour

Maximum pressure -96.06pa
Minimum pressure -0pa

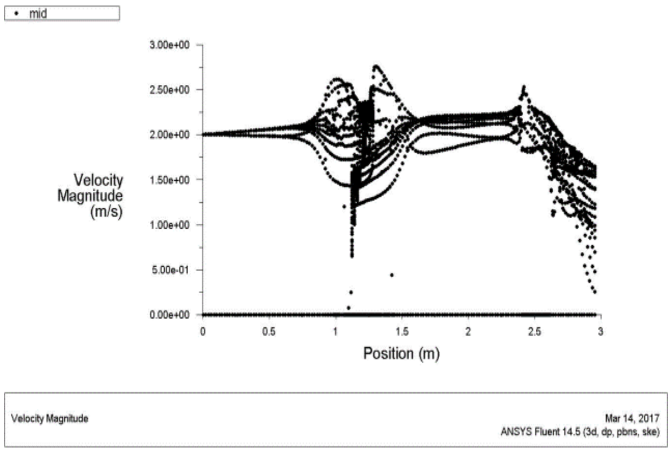

Fig.15. Velocity contour

Maximum velocity -2m/s
Minimum velocity -1.5m/s

VI. CONCLUSION

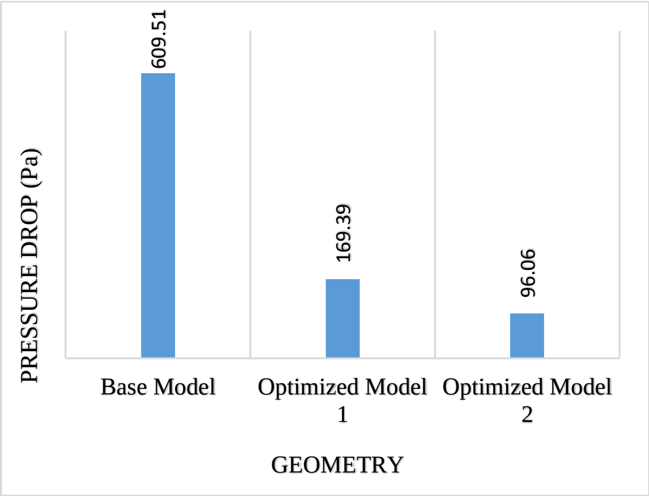

Fig.16. Static Pressure Drop Difference

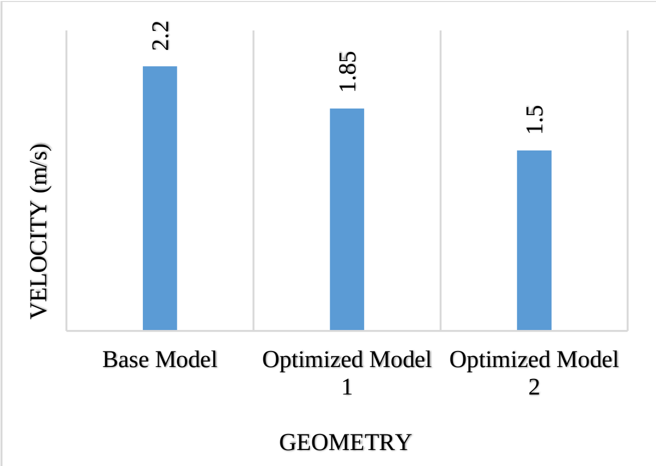

Fig.17. Velocity at outlet

From the graphs obtained above we can clearly conclude that the second optimized design is very effective One in reducing pressure drop and to enhance effective Velocity distribution with less cost of manufacturing. In first optimization the pressure drop is reduced to 169.39pa and the velocity distributed is 1.83m/s. The velocity distribution which we have achieved has its outlet velocity of 1.5m/s. Hence From the above analysis we can conclude that the second optimized design is highly effective and it is ensured by CFD analysis.

REFERENCES

[1] Dr. Neihad Al-Khalidy, Design optimization of industrial ducts using CFD, Third international conference, Australia, December 2015, PP (1-60).

[2] Malgorzat Wiatros-Motyka, Optimizing fuel flow in pulverized coal and biomass-fired boilers, IEA Clean Coal Centre, United Kingdom, January 2016, PP (1-61).

[3] Naveen T.K.Chelliah A, Analysis and Efficiency Enhancement of Mettur Power Station, Indian Journal of Science, 2015, PP (1-5).

[4] Indrusiak, M. L. S, Beskow A. B and Da Silva.C.V, Thermal Power Plant Boiler Misoperation using CFD, UNISINOS, Brazil, PP (1-4).

[5] JurgenGrasel, Manuel Pierre, Jacques Demolis, Turbomeca, Parametric Interturbine Duct Design and Optimization, 25th international congress, America, PP (1-11).

[6] Jeffrey D. Spitler, Ph.D., P.E, Optimum Duct Design for Variable Air Volume Systems, International journal of American society of heating, PP (1-24).

[7] Ramesh Avvari, Dr. Srinivas Jayanti and S. Gowrisankar, Shape Optimization of Power plant Ducting Using CFD, ASME International Mechanical Engineering Congress, USA 31-10-2014, PP (1-6).

[8] G. Alessi1, L. Koloszar1, T. Verstraete1, B. Blocken and J.van Beeck, Adjoint Flow optimization of U-Bend duct, ECCOMAS Thematic Conference - CFD & Optimization, Antalya, 2014, PP (1-4).

[9] Athanasios Tzanakis, Duct optimization using CFD software ANSYS Fluent Adjoint Solver, Chalmers University Of Technology, Sweden 2014, PP (1-4).

vi