Journal Homepage: -www.journalijar.com

INTERNATIONAL JOURNAL OF ADVANCED RESEARCH (IJAR)

Article DOI:10.21474/IJAR01/20787
DOI URL: http://dx.doi.org/10.21474/IJAR01/20787

RESEARCH ARTICLE

MODELING AND ANALYSIS OF PRESSURE DROP IN FLUE GAS DUCT PATH USING COMPUTATIONAL FLUID DYNAMICS

Hemex Premjiyani[1], Arunesh Dwivedi[2] and Hiren Rana[3]

1. Faculty of Technology and Engineering, The Maharaja Sayajirao University, Vadodara,Gujarat, 390001, India.
2. Accredited Energy Auditor, Electrical Research & Development Association, Vadodara, Gujarat, 390010, India.
3. Faculty of Technology and Engineering, The Maharaja Sayajirao University, Vadodara, Gujarat, 390001, India.

Manuscript Info

Manuscript History
Received: 15 February 2025
Final Accepted: 19 March 2025
Published: April 2025

Abstract

Nowadays, in contrast to the worldwide energy crisis and the global commitment to mitigate carbon footprints from the source, energy conservation and carbon emission reduction are vital and mandatory for almost all entities; hence, so for coal-based thermal power stations. Higher auxiliary power consumption than the optimum value is observed in almost all power plant auxiliaries, starting from the boiler feed pump to the induced draft fans during the energy audit. Also, the oldness of the power plant worsens this scenario of high auxiliary power consumption. Hence, any modification in the design of rotary machines, rerouting of ducts and pipelines, and resizing of equipment like fans, pumps, blowers, and flow paths will help and add an advantage to the energy-saving program. In this paper, the author is investigating the effect of baffles (guide plates) at bend locations in the flue gas path to streamline the flue gas flow and reduce pressure drop. This paper focuses on the analysis of flue gas duct pressure reduction using baffles and comparing it with a no-baffles case. For analysis purposes, one location at a 90-degree bend has been chosen, and the model was constructed with the help of the ANSYS Fluent solver. The design of the 3D model was modeled in the geometry software ANSYS SpaceClaim 2024 R1, and the analysis was done in the CFD ANSYS Fluent 2024 R1 solver.

Corresponding Author:-Hemex Premjiyani
Address:-Faculty of Technology and Engineering, The Maharaja Sayajirao University, Vadodara, Gujarat, 390001, India.

……………………………………………………………………………………………………....

Introduction:-

In thermal power plants, steam is generated in the steam generator. Thermal energy is supplied by the combustion of coal in the furnace zone. The byproduct of the combustion process is hot flue gas, which is transported via duct and finally exhausted from the chimney. The flue gas duct, heat recovery equipment (an air heater), and pollution control equipment (an electrostatic precipitator) are parts of the flue gas exhaust system. Energy to extract boiler flue gas from the furnace zone comes from the operation of an induced draft fan, which is electrically operated. Energy consumed to extract and discharge flue gas into the atmosphere is proportional to the rate of flow of flue gas and the magnitude of the draft generated. Longer length of duct, control dampers, APH, ESP, and bends in the flow path added restriction to flue gas flow. A certain draft pressure is also maintained in the furnace by extracting flue gas at a certain rate. All these factors and considerations were added to the design of ID fans operating under draft pressure. The electrical energy needed to overcome these restrictions and requirements for certain drafts is supplied by the ID fan. Turbulence and eddy generation at the flue gas duct bend location are added to the extra suction pressure requirement of the fan and vice versa electrical energy requirement. Making the flue gas path more efficient will reduce the ID power consumption to some extent. In this paper, the author, using ANSYS software, investigates and evaluates the pressure loss at a 90-degree bend location without baffles and after providing two sets of baffles (guide plates). Comparing the results of the pressure drop will give the magnitude of the reduction.

FIGURE 1.1 Boiler Duct Outer View

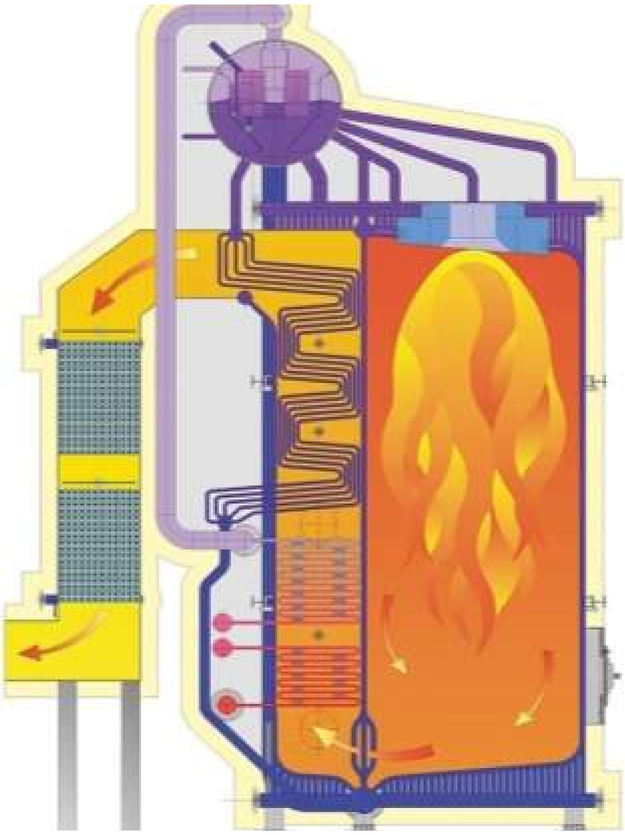


FIGURE1.2 Schematic Diagram of Internal Parts of the Boiler and Flue Gas Path

(Source: www.outselluar.com)

BAFFLES (FLOW GUIDE PLATE)

Baffles are flow-directing vanes or panels used in some industrial plants and power plant boiler ducts. Baffles are used to streamline the fluid flow and reduce the internal circulation of gas at bend locations, thus decreasing the pressure drop at 90° bends in the boiler duct. The distance between adjacent baffles is called baffle spacing. Baffles are held in position by means of baffle spacers or welded at the side of the duct.
In this project, a finite element analysis of the boiler duct was carried out to validate the design for actual working conditions. The main tasks involved in the project are performing the 3D modeling of the boiler duct and finite element analysis.

OBJECTIVE AND METHODOLOGY

PROBLEM DESCRIPTION

The objective of this project is to make a 3D model of the flue gas duct and study the CFD and fluent behavior of the flue gas duct by performing finite element analysis. 3D modeling software (PRO-Engineer) was used for designing, and analysis software (ANSYS) was used for CFD analysis.
The methodology followed in the project is as follows:

- Create a 3D model of the flue gas duct assembly using geometry software. ANSYS SpaceClaim 2024 R1
- Convert the surface model into ANSYS Mechanical 2024 R1 and do analysis.
- Performance Fluent Analysis on the ANSYS Fluent 2024 R1.
- Perform CFD analysis on the existing model of the surface flue gas duct for pressure inlet and mass flow outlet conditions applied, and find out the pressure outlet.

PLANT OVERVIEW

Duct dimensions and other relevant parameters of a typical 210 MW flue gas duct are taken for analysis and modeling. The flue gas duct and design data have been shown in the table below.

TABLE I. Steam Boiler Design Data

SR.NO.	PARAMETER	UNIT	VALUE
I.	Steam Boiler Make	BHEL	
II.	Plant Generation Rating	MW	210
III.	Boiler Duct Size(l, W, h)	m	50, 2.90, 2.65
IV.	Cross sectional area	m^2	7.685
V.	Mass flow rate single duct	kg/sec	118
VI.	Temperature	°C	165
VII.	Density of FG	kg/m^3	0.82
VIII.	Velocity	m/s	18.7
IX.	*Inlet Pressure	Pa (g)	2746

***Remark:** The actual pressure inside the duct is a negative draft. For analysis purposes, the author assumes positive pressure to estimate the pressure drop across the duct using ANSYS modeling for simplicity and better understanding. The resulting pressure drop is a good approximation of the actual phenomenon and does not change the objective of the analysis.

FLUE GAS DUCT MODELING AND ANALYSIS WITHOUT BAFFLES

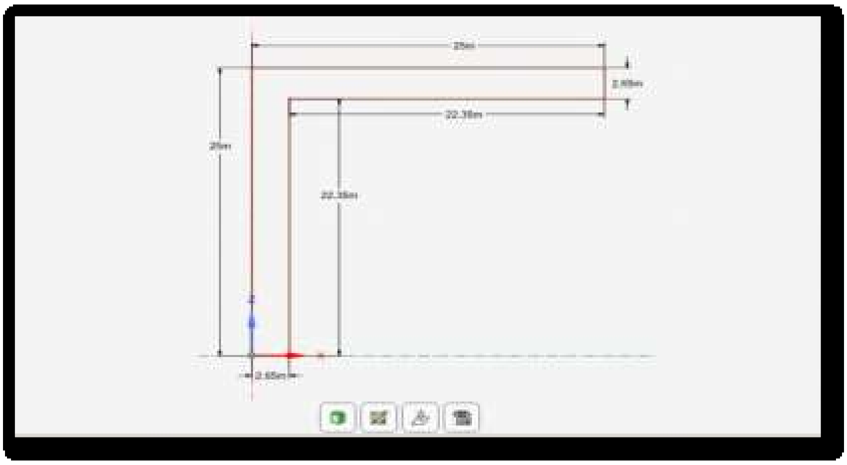
The following parameters are inserted in ANSYS and followed by the procedure below:

ANSYS → workbench→ select analysis system → fluid flow fluent → double click

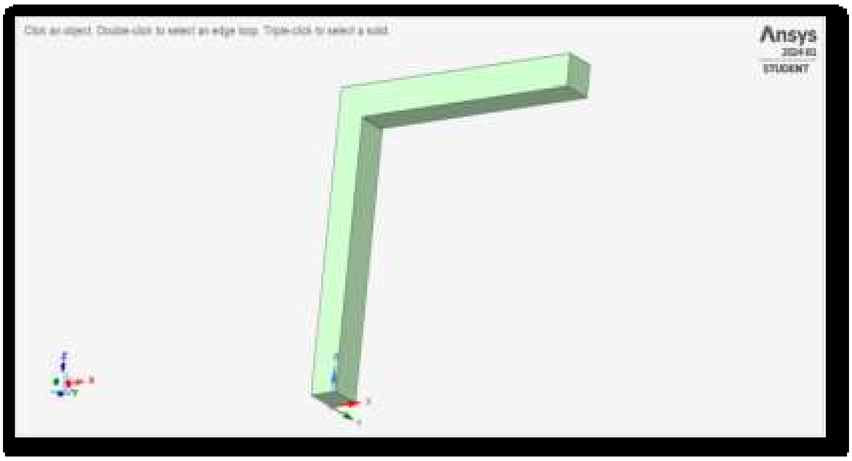
Step 1: SpaceClaim 2024 R1 Used for Modeling of Flue Gas Duct

Select geometry → right click → design geometry → ok

Flue Gas Duct 2D Diagram:



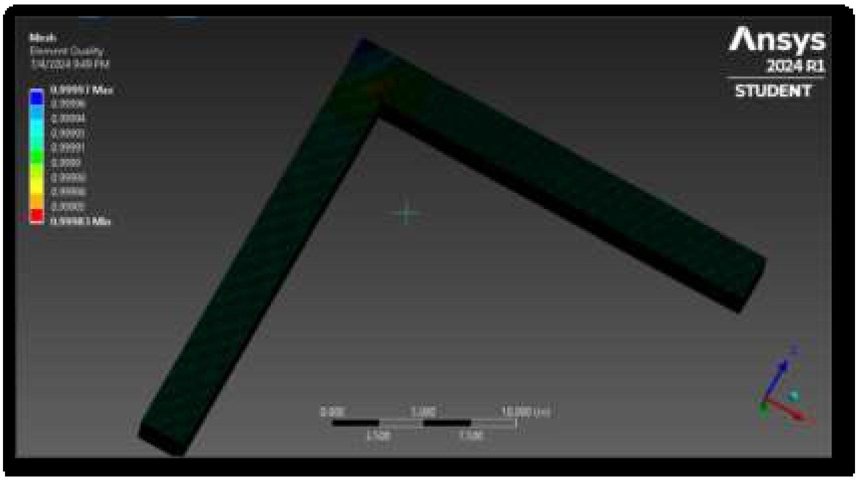
Flue Gas Duct 3D Diagram:



The model is designed with the help of ANSYS SpaceClaim.

Step 2: Mechanical 2024 R1 Used for Flue Gas Duct Meshing and Sectionalized

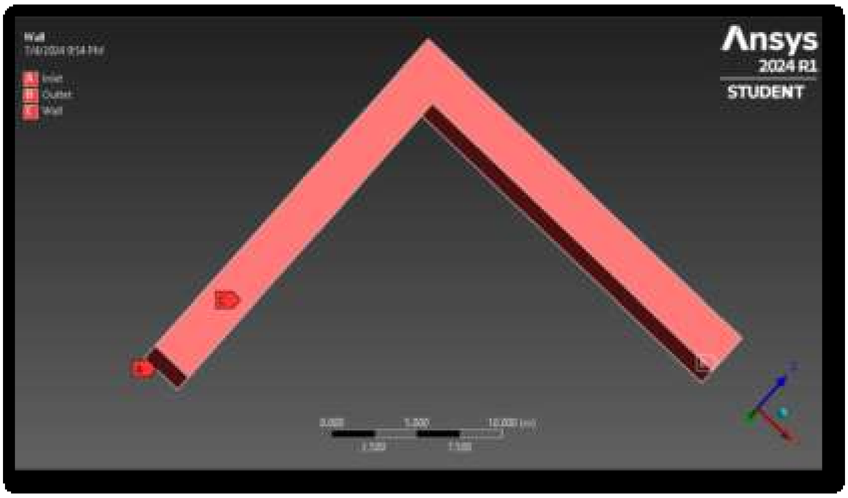
Select mesh on the workbench → right-click →edit → Select mesh on the left side of the part tree → right-click → generate mesh.



Select faces → right-click → create named section → enter name → inlet

Select faces → right-click → create named section → enter name → outlet

Select faces → right-click → create named section → enter name → wall



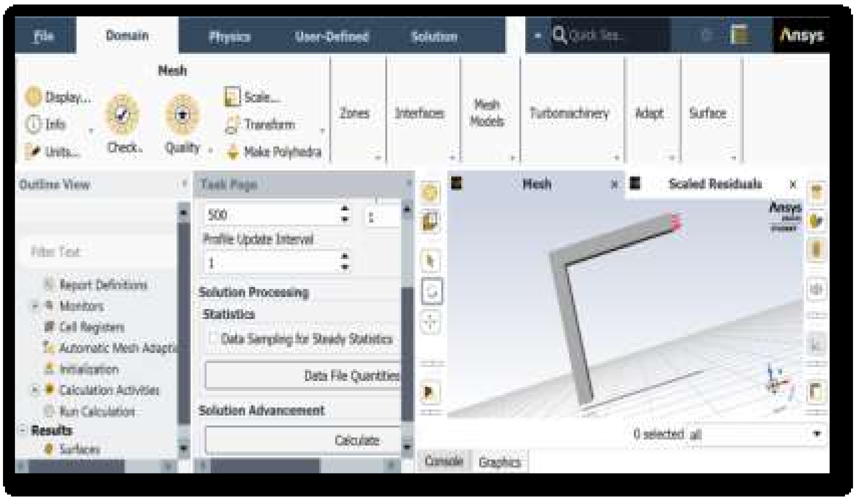
The model was meshed and sectionalized in ANSYS Mechanical. Then all thickness edges are meshed with 360 intervals. A tetrahedral mesh is used. So the total number of nodes and elements is 1025164 and 972360.

Step3: Fluent 2024 R1 Solver-Based Problem Solving

The procedure adopted is given below:

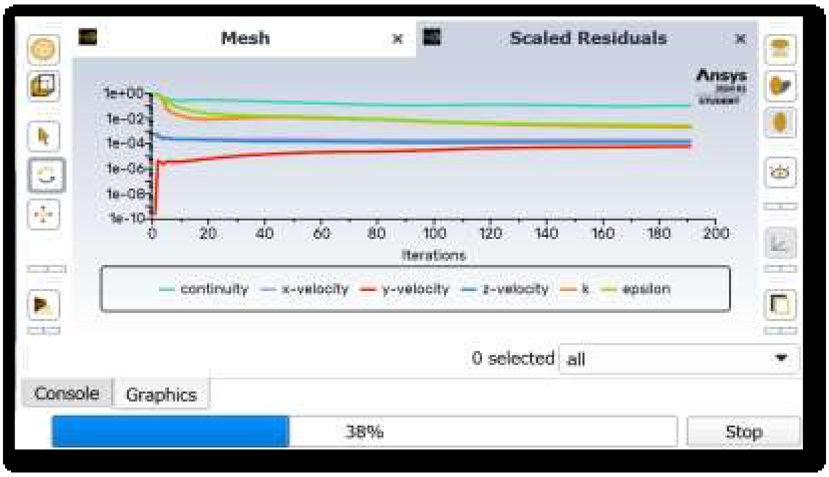
Model → Viscous → edit → k-epsilon → Standard → ok

Select air boundary conditions → select air inlet → Edit → Enter air mass flow rate → 118 kg/s → Inlet pressure → 2746 Pascal (g) → ok



Initialization → Hybrid initialization →done

Run calculations → no. of iterations = 500 → Calculate → Calculation complete

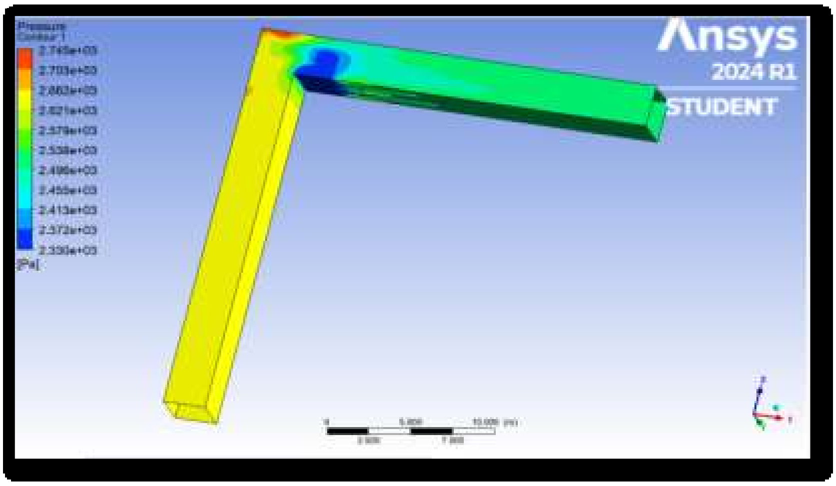


ANSYS Fluent Solver is used for solving problems. The analysis by ANSYS Fluent CFD is used in order to calculate pressure, velocity, and mass flow rate.

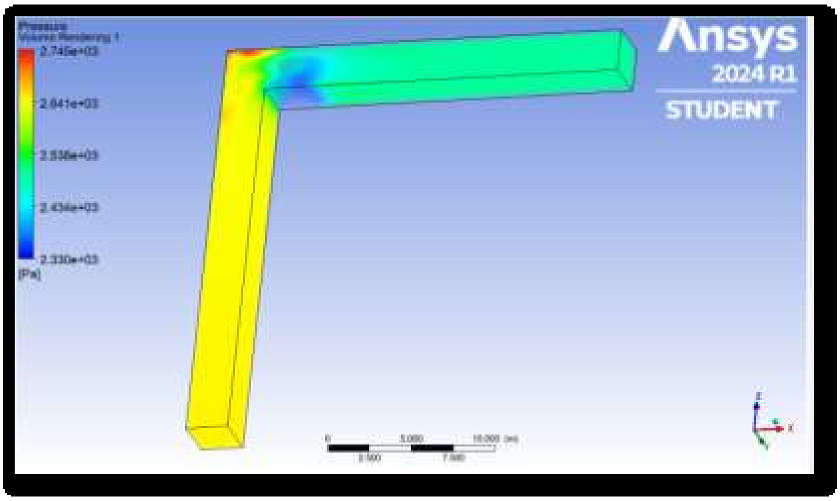
Step4: CFD Post 2024 R1 Used for Result Viewing

Results → Graphics and animations → Volume rendering → Setup

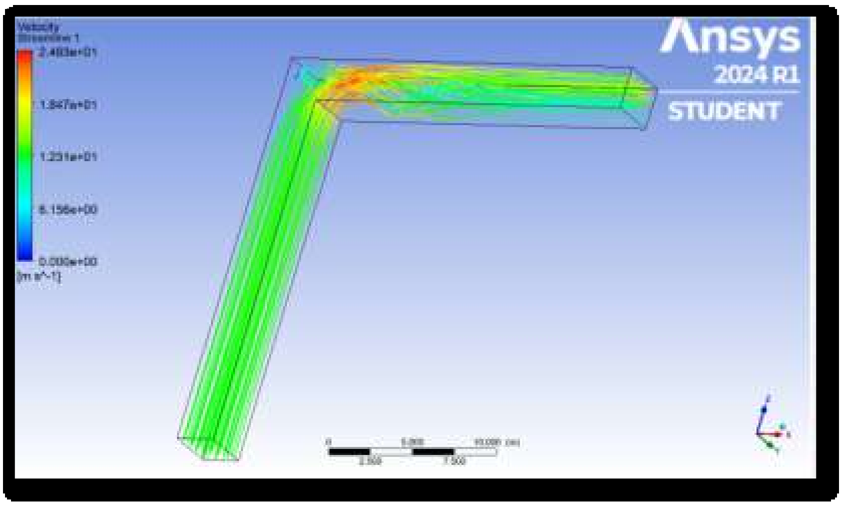
Pressure Contour View



Pressure Volume View

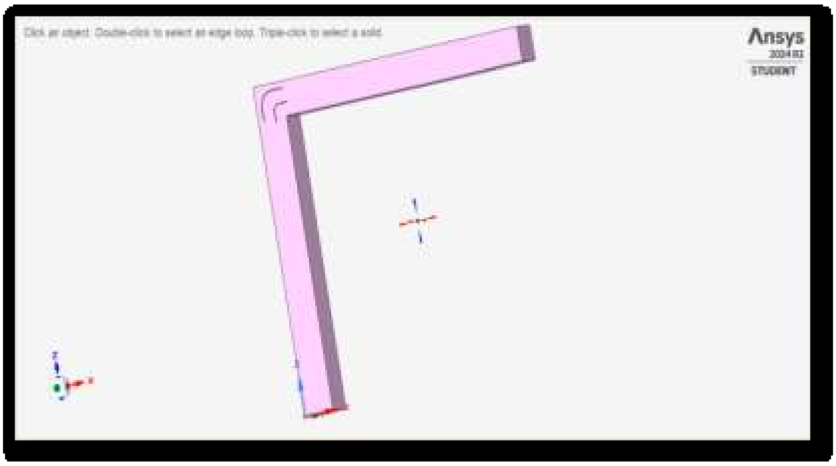


Velocity Stream Line

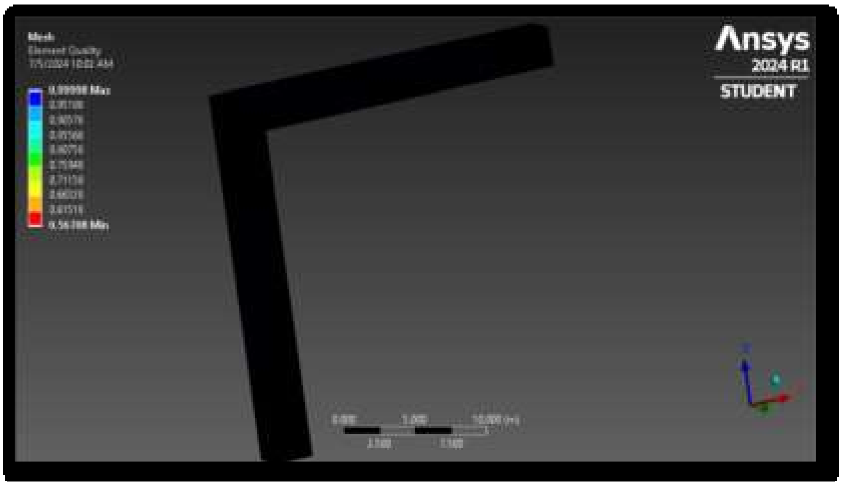


FLUE GAS DUCT MODELING AND ANALYSIS WITH BAFFLES

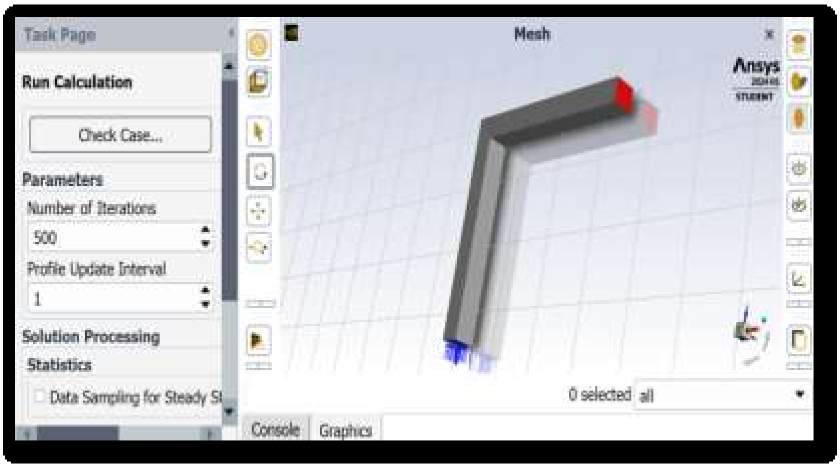
Step1: Modeling of Flue Gas Duct with Baffles



Step2: Meshing of Flue Gas Duct with Baffles Model

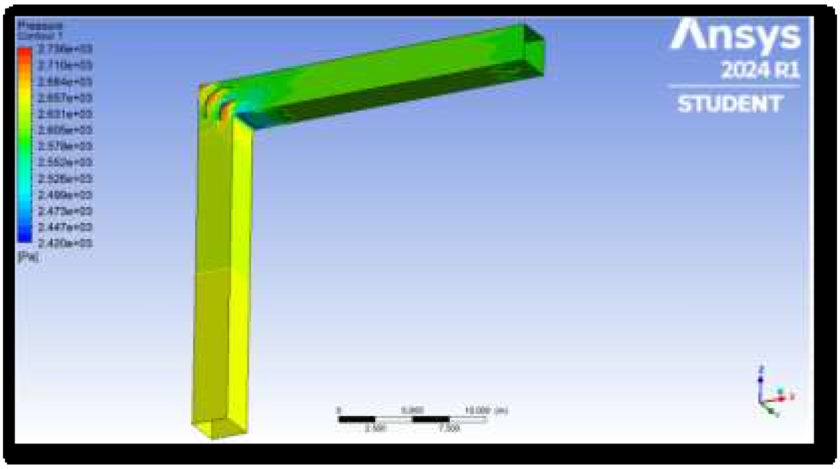


Step3: Fluent in Solving Problems Flue Gas Duct with Baffles Model

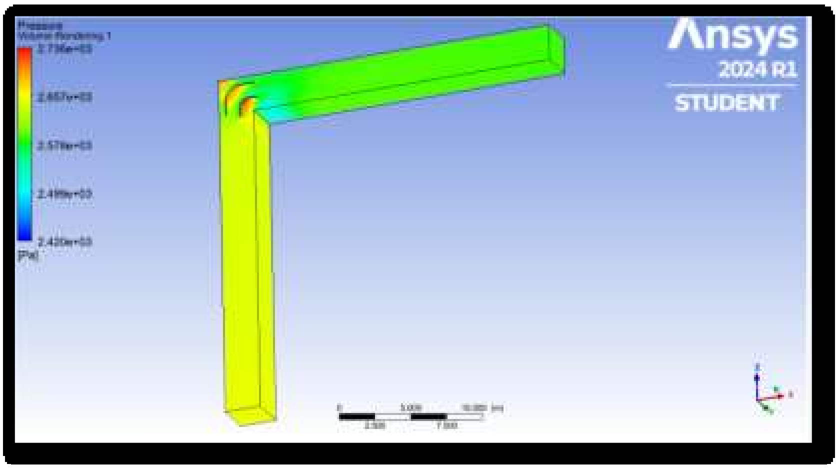


Step4: CFD Post 2024 R1 Used for Result Viewing

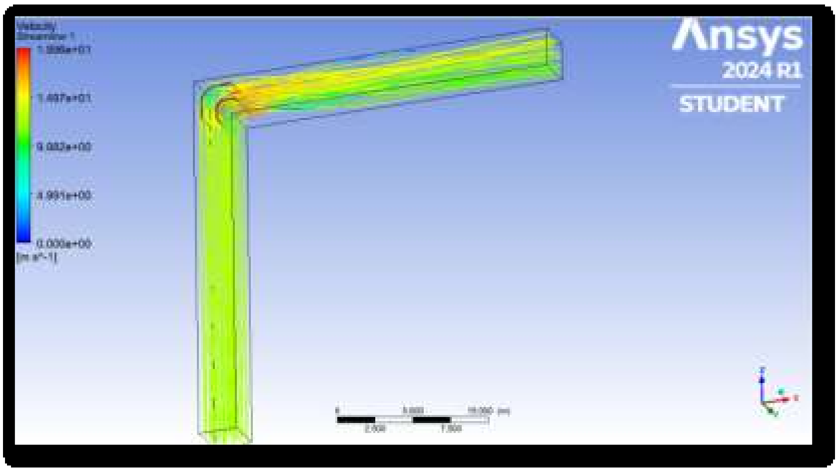
Pressure Contour View



Pressure Volume View



Velocity Streamline line



RESULTS AND DISCUSSION

Technical Result of Analysis Based on FLUENT Solver

The results obtained for flue gas duct pressure drop analysis are summarized below with and without baffles:

Duct without Baffles			
Zone	Velocity	Pressure	Mass Flow Rate
Inlet	1.253e+1 [m s^-1]	2.650e+3 [Pa]	7.969e-2 [kg s^-1]
Outlet	1.253e+1 [m s^-1]	2.498e+3 [Pa]	-7.973e-2 [kg s^-1]

Duct with Baffles			
Zone	Velocity	Pressure	Mass Flow
Inlet	1.253e+1 [m s^-1]	2.650e+3 [Pa]	7.973e-2 [kg s^-1]
Outlet	1.254e+1 [m s^-1]	2.590e+3 [Pa]	-7.973e-2 [kg s^-1]

The pressure drop of the flue gas duct without baffles is 152 Pascal, whereas with two parallel baffles at a 90-degree bend, it is 60 Pascal. This has a clear reduction in flue gas pressure drop of 92 Pascal, which is equal to 9.38 mmWC.

Considering the four numbers of bends normally present in any power plant flue gas path, the reduction in pressure drop will be9.38×4 = 37.52 mmWC.

The mass flow rate is consistent with the value of 118 kg/sec.

Volumetric flow considering flue gas density of 0.82 kg/m³ at 160 °C =118 /0.820 = 143.90 m³/sec

For the interest of power plant engineers, the author calculated the energy and monetary savings incurred with a reduced pressure drop using baffles in the flue gas duct.

Now, we calculated fan power savings in kW.

Considering a fan and motor combined efficiency of 60%.

Fan Output $=\frac{\boldsymbol{Volumatricflow(m^3/s)\times Pressure(mmwc)}}{\mathbf{102\ \times Overall\ efficiency}}$kW

$= \frac{\mathbf{143.90\times 37.52}}{\mathbf{102\times 0.6}}$kW

= 87 kW

So the power saving of the fan is 87 kW when the flue gas duct has baffles in all four locations at a 90° bend.

Economic Analysis of Flue Gas Ducts WITH BAFFLES

A cost-economic analysis is required to check the viability of the proposal. The payback period is one of the simplest investment appraisal techniques. This is defined as the time in which the initial cash outflow of an investment is expected to be recovered from the cash inflows generated by the investment. It is estimated that the total project cost for retrofitting and installing the two numbers of baffles at the four-bend location in the flue gas path is approximately Rs. 4 lakhs. This is inclusive of all costs, including labor and installation costs.

Net power savings have been calculated at 689 MWh, considering plant operation for 330 days in a year. The estimated power savings per annum is approximately Rs. 22.00 lakhs per year, considering the electricity cost of the power plant at Rs. 3.2 per kW. The estimated simple payback period is within 3 months.

CONCLUSION

The pressure drop optimization in the flue gas duct at sudden bend locations was carried out using the ANSYS CFD computational tool. Actual duct dimensions and operating parameters of a coal-based thermal power plant of 210 MW capacity are used for simulation and analysis. Flue gas flow in one pass is modeled after a two-pass boiler exhaust flue gas system. The flue gas flow rate is halved from the total value and used for analysis.

The numerical computations were done using continuity, momentum, and k-ε turbulence kinetic energy equations at discretized zones of the domain. The analysis focuses on reducing the pressure drop in the duct using a 90° circular baffle at a single-bend location. The difference in pressure drop values is obtained without and with the baffle condition, which is satisfactory. The resulting pressure drop value is lower in the baffle case as compared to a duct without a baffle. So the author concludes that the flue gas duct with baffles is more energy efficient, with 87 kW of electrical power savings.

BIOGRAPHICAL DETAILS OF THE AUTHORS

Hemex Premjiyani is a student of the Faculty of Technology and Engineering at the Maharaja Sayajirao University (MSU), Vadodara. He is doing an M.Tech in Electrical Power Engineering at the Maharaja Sayajirao University (Vadodara), and his project is with the Electrical Research & Development Association (ERDA), Vadodara,on Energy Audit Standardization of power plant and process industries, and BE Electrical Engineering from Government Engineering College, Palanpur. His professional interests include thermodynamics, process optimization, heat and mass transfer, power systems, high-voltage engineering, switchgear and protection, and energy conservation. He has worked at the 66 kV GETCO Substation (Gujarat Energy Transmission Corporation Limited) and the four-MW solar power plant for approximately 1 year.

Arunesh Dwivedi is Deputy Manager at the Electrical Research and Development Association (ERDA), Vadodara. He obtained his M.Tech. honors in energy management from the School of Energy and Environmental Studies (DAVV Indore) and his BE in chemical engineering from the National Institute of Technology, Raipur. He is an accredited energy auditor from the Bureau of Energy Efficiency, Government of India. He has been working in the energy audit and power plant performance field for the last 17 years and has conducted more than 250 energy and performance audits of various process industries, the cement sector, and power plants. His professional interests include thermodynamics, fluid dynamics, transport phenomena, heat and mass transfer, process optimization, and energy conservation. He has also worked on the process and energy optimization of cement plants.

REFERENCES

1. M. Sai Krishna, Dr. E. subbarao & Kiran Kumar, “Design and Thermal Analysis of Steam Boiler (Without & With Baffles) Used in Small Power Plants,” International Journal of Advanced Research in Science, Engineering and Technology, 5(5), ISSN: 2350-0328, May 2018.
2. Jeanaustin .S., “CFD Analysis for Layout Optimization of Flue Gas Ducting Between Air Preheater and Electrostatic Precipitator,” International Journal of Engineering Development and Research, 6(3), 2321-9939, 2018.
3. www. outselluar.com