CFD STUDY ON FLOW THROUGH SINGLE-HOLED ORIFICE PLATES IN SERIES

Akash S.[1], Gowsihan N.[1], Hrithik C.[1], Dr. Vivek Vitankar [2]

[1] Chemical Engineering 2018-2022, School of Chemical and Biotechnology (SCBT), SASTRA University;

[2] Director, FluiDimensions

Abstract

Flue gas ducting networks in coal power plants are critical pathways for transporting combustion off-gases through treatment equipment such as air preheaters, electrostatic precipitators, and fans. Common operational challenges in such systems include high pressure drop, duct erosion, and unequal mass flow distribution across outlets, all of which raise operating costs and reduce equipment life. This study addresses a real-world industrial problem involving the ducting network between the air preheater and electrostatic precipitator in a coal power plant. Full-scale 3D steady-state CFD simulations were performed using ANSYS Fluent 2021 R1, employing the k-ε realizable turbulence model with an Eulerian-Lagrangian approach for the solid ash discrete phase. The original geometry was analysed to identify regions of high pressure drop, flow separation, and erosion. Four progressively refined design modifications incorporating fillets, V-shaped baffles, expansion joints, and arc-radius baffles at critical junctions were evaluated and compared against the original geometry. The final proposed design achieved a pressure drop reduction of 72%, improved mass flow uniformity across all four outlets to within ±10% deviation and reduced monthly pumping costs by 72%. This study demonstrates that targeted minor geometric modifications, guided by CFD analysis, can deliver substantial operational and economic improvements in industrial ducting systems without requiring major plant layout changes.

Keywords: CFD; ANSYS Fluent; Pressure Drop; Erosion; Ducting Network; Operating Cost