

Numerical Study Of Air Flow Efficiency In Filter Assemblies Using Computational Fluid Dynamic

Khairul Faizal bin Abdul Hamid^{1*}, Amizan bin Abdullah², Farah Sofhia binti Mohd Zin³

¹Department of Mathematics, Science and Computer, Politeknik Sultan Mizan Zainal Abidin, 23000 Dungun, Terengganu

²Department of Mathematics, Science and Computer, Politeknik Mukah, 96400 Mukah, Sarawak

³Department of Civil Engineering, Politeknik Sultan Mizan Zainal Abidin, 23000 Dungun, Terengganu

***Corresponding author E-mail:** khairulfaizal@psmza.edu.my

Abstract

This study investigates airflow efficiency in air filter assemblies through a computational fluid dynamics (CFD) simulation using ANSYS CFX. The primary objective is to evaluate the pressure drop characteristics of a digitally modeled Basic filter design and compare the simulation results with real-world performance data from the Salutory Avenue filter, a commercially available product used in gas turbine intake systems. The CFD simulation was conducted only on the Basic model, which was created using SolidWorks and analyzed in ANSYS Workbench (Academic Edition) under various inlet velocities (0.032, 0.063, 0.094, and 0.126 m/s). The simulation applied structured meshing, appropriate boundary conditions, and a standard k- ϵ turbulence model, with convergence criteria set to 1000 iterations and 1×10^{-4} RMS. Post-processing focused on pressure vector differences. The results showed a consistent trend with the real filter data, where pressure drops increased with higher inlet velocities. Although some variations in magnitude were noted, the CFD results demonstrated coherence with the field data, validating the Basic model's accuracy in predicting airflow resistance. This approach supports the use of CFD as a reliable predictive tool in preliminary filter design and optimization for air intake systems.

Key Words: *Filter, Computational Fluid Dynamics (CFD), Air Filter, Pressure Drop, Gas Turbine, ANSYS CFX, Filter Performance*

1.0 Introduction

Gas turbine power plants rely critically on clean intake air to maintain efficiency and reliability. Filters installed in the intake system remove particles and contaminants, preventing erosion, fouling, and loss of performance in compressor stages (Effiom et al., 2024). However, filtration introduces a pressure drop, which directly reduces turbine efficiency, power output, and increases fuel consumption (Mohammed et al., 2021). Managing this trade-off between filtration effectiveness and aerodynamic resistance is thus essential.

Computational fluid dynamics (CFD) has become an indispensable tool for the design and optimization of gas turbine inlet filters. It allows precise evaluation of pressure drop, flow uniformity, and turbulence effects in complex geometries before physical prototyping (Abam, et al., 2016). For example, Abam et al. (2016) used CFD to assess a multi-stage filter housing for a heavy-duty industrial turbine, finding acceptable total pressure drops and satisfactory flow distribution. Similarly, Conical cartridge filters have been optimized using flows and cleaning strategies based on CFD analyses (Yi, J et al., 2023).

Experimental work has also reinforced the significance of filter pressure drop in real-world turbine operation. A study of offshore gas turbines showed that incremental filter pressure losses of just 1 Pa increased fuel costs by ~RM 41 per hour, and predicted filter life of only ~41 months before reaching allowable pressure limits (Effiom et al., 2024). Another investigation showed that stage-wise filter selection in turbine intake systems can lead to up to 2.5 % loss in shaft power for a 1000 Pa intake loss (Jin et al., 2021).

Despite these advances, detailed CFD comparisons between alternative filter geometries—quantifying how different designs perform under varying inlet velocities—remain limited. To

address this gap, the present study uses ANSYS CFX to simulate two filter configurations, **Salutary Avenue** and **Basic**, subject to inlet velocities of 0.032–0.126 m/s. Pressure drop measurements are compared and analyzed to guide filter design optimization.

2.0 Methodology

In this study, computational fluid dynamics (CFD) was used to evaluate the airflow characteristics and pressure drop of a digital filter model referred to as the **Basic** design. The simulation results were compared to field performance data collected from a commercially available filter, known as the **Salutary Avenue**, which had been installed in an operational gas turbine air intake system. This comparison aimed to assess the accuracy and effectiveness of using CFD to predict filtration performance in real-world conditions. Simplify process to setup a simulation in CFD software illustrate in the flow chart shown in Figure 1.

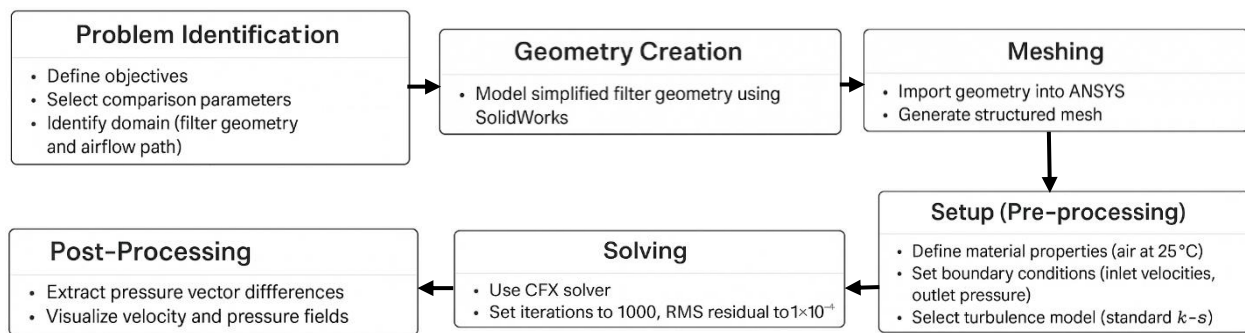


Figure 1. Flow chart of methodology

The geometry of the Basic filter model was developed using SolidWorks, focusing on simplified representations of the airflow path and filtration zones to enable manageable computational analysis as shown in Figure 2. This digital model was then imported into ANSYS Workbench Academic Edition, and simulations were performed using the CFX module. The CFD analysis was conducted under the assumption of steady-state, incompressible, and isothermal air flow, with air properties set at 25 °C (density: 1.184 kg/m³; viscosity: 1.849×10⁻⁵ Pa·s).

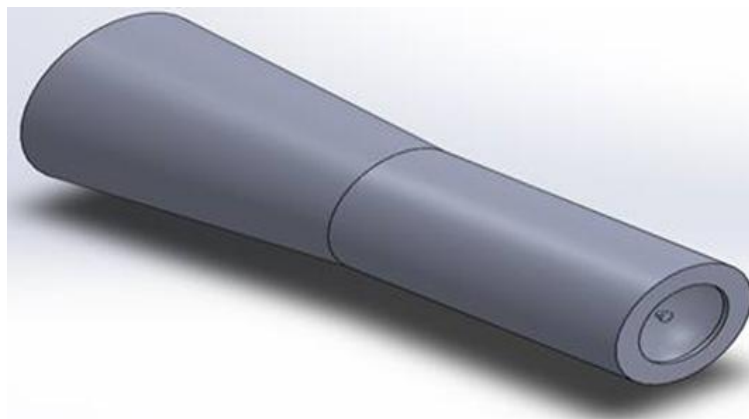


Figure 2. Filter geometry using SolidWork.

A structured mesh was generated using ANSYS Meshing, with increased resolution near regions where pressure gradients were expected to be highest. Meshing sizing between 8mm to

11mm selected to produce a mesh mode shown in Figure 3. The mesh quality was verified to ensure numerical stability and convergence of the simulation such sizing is set to be between 8mm and 11, detail of mesh setting show in Figure 4. Boundary conditions for the analysis included four discrete inlet velocities: 0.032, 0.063, 0.094, and 0.126 m/s. The outlet were assigned a gauge pressure of 30 kPa, and no-slip wall conditions were applied to all solid surfaces.

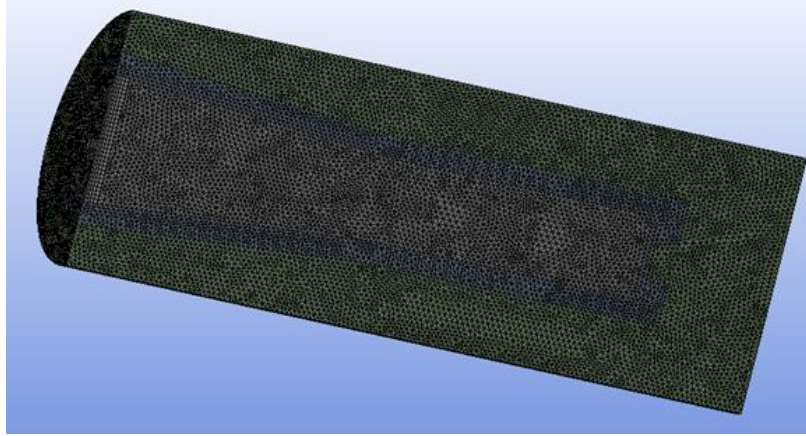


Figure 3. Meshing Model with sizing 8mm to 11mm

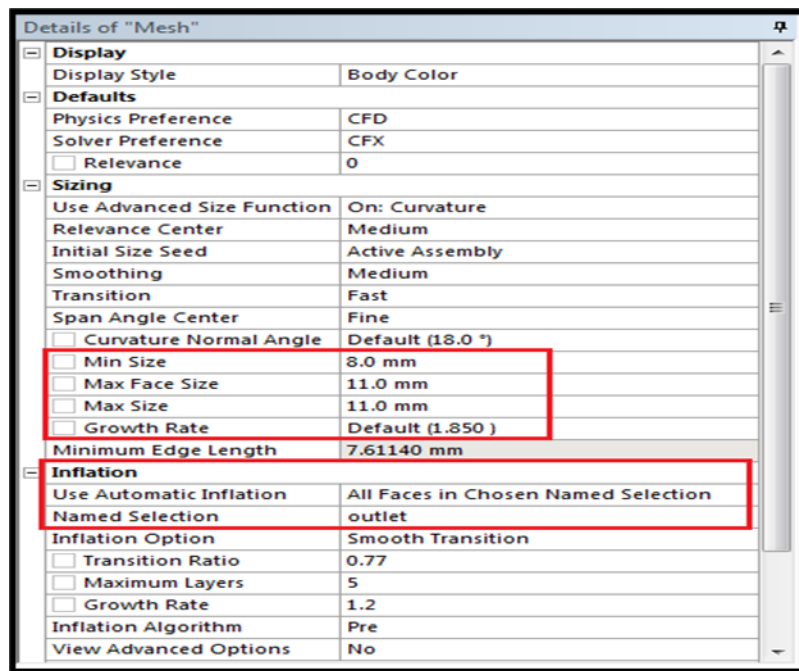


Figure 4. Sizing and inflation setting

The simulation solver was configured in CFX using a pressure-based, steady-state approach. The standard k- ϵ turbulence model was selected to account for flow fluctuations and boundary layer effects. A maximum of 1000 iterations was allowed, and convergence was defined by a root mean square (RMS) residual target of 1×10^{-4} . Post-processing was conducted using CFX-Post, with the primary focus on evaluating the pressure vector difference across the filter media. In the pre-processing setup, the filter domain will be set as shown in the Table 1.

Table 1: Filter Domain Setup

Setting	Details
Analysis type	Steady state
Domain type	Porous domain
Fluid definition	Air at 25°C Continuous fluid
Particles definitions	Air at 25°C Particles transport fluid Uniform in Diameter by number (0.3-1micron)
Solid definitions	Material (polystyrene) Continuous solid
Fluid Models	Isothermal (297K) Turbulence model (k-epsilon)

To validate the simulation, the CFD results for the Basic model were compared with actual pressure drop measurements obtained from the Salutary Avenue filter during field operation under matching inlet velocity conditions. This comparative approach provided insight into the reliability of CFD simulations in predicting the performance of gas turbine intake filtration systems.

3.0 Results

The simulation of airflow through the Basic filter geometry was performed at four inlet velocities: 0.032, 0.063, 0.094, and 0.126 m/s. For example, Figure 5 show how the velocity streamline at 0.032m/s. The primary parameter of interest was the pressure drop across the filter, calculated as the difference between the inlet and outlet static pressure. The simulated values were then compared with actual pressure drop data from the Salutary Avenue filter; a commercially deployed product used in gas turbine intake systems. The comparison is summarized in Table 2.

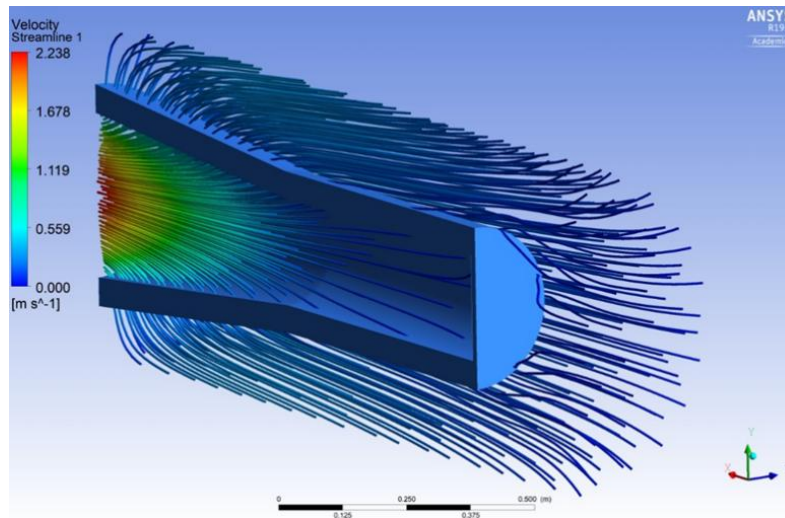
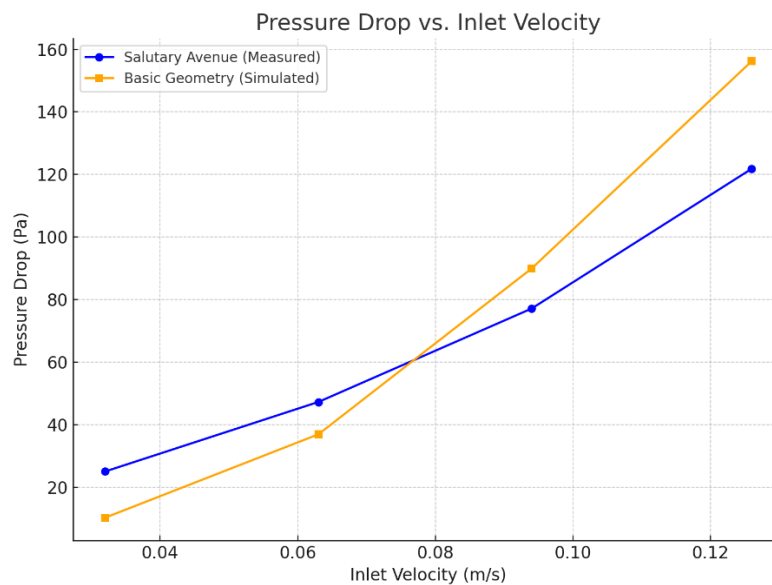


Figure 5. Velocity streamline of basic geometry at 0.032 m/s

Table 2. Pressure Drop Comparison Between Salutory Avenue Filter and Basic Geometry Model

Velocity (m/s)	Salutory avenue	Basic geometry		
	Pressure drops (Pa)	Inlet (Pa)	Outlet (Pa)	Pressure drops (Pa)
0.032	25	30009.738	29999.514	10.224
0.063	47.23	30035.242	29998.361	36.881
0.094	77.1	30085.338	29995.434	89.904
0.126	121.81	30148.1	29991.893	156.207

**Figure 6.** Pressure Drop vs. Inlet Velocity for Salutory Avenue and Basic Geometry Filter

As shown in Table 2 and Figure 6, both the simulated (Basic Geometry) and measured (Salutory Avenue) pressure drops exhibit a clear upward trend with increasing inlet velocity. At the lowest inlet velocity (0.032 m/s), the pressure drop in the Basic model was 10.224 Pa, noticeably lower than the 25.00 Pa recorded from the real filter. As the velocity increased to 0.126 m/s, the pressure drops for the Basic model rose to 156.207 Pa, closely approaching the Salutory Avenue's 121.81 Pa. Both datasets exhibit a strong positive correlation, likely above 0.95, as the pressure drop consistently increases with inlet velocity, though the simulated data shows a slightly higher rate of increase. The trendlines, inferred from the plotted points, suggest a reliable linear relationship, with the measured data slightly underperforming compared to the simulation at higher velocities, possibly due to real-world variables not accounted for in the simulation.

These results indicate that while the Basic geometry underestimates the pressure drop at lower flow rates, it closely approximates real-world behaviour at higher velocities. This convergence supports the validity of using CFD to simulate airflow characteristics in preliminary filter design, particularly for estimating high-velocity performance.

4.0 Conclusion

This study demonstrates the effectiveness of computational fluid dynamics (CFD) simulation in evaluating the performance of a digital air filter design compared to an existing commercial product used in gas turbine applications. Using ANSYS CFX, the basic geometry model was developed and simulated across a range of inlet velocities. The results, particularly the pressure drop values, showed a consistent trend with those obtained from the real-world performance of the Salutory Avenue filter.

The pressure drops for both filter types increased with inlet velocity, reflecting realistic fluid dynamic behaviour. Notably, the simulated basic geometry yielded pressure drop values that were in close agreement with field data from the commercial Salutory Avenue filter, indicating the accuracy and reliability of the CFD approach.

This coherence between simulated and real data supports the viability of using CFD tools in the early-stage design and optimization of air filter assemblies. It enables designers to predict performance outcomes and make informed design decisions without excessive reliance on physical prototyping. Thus, CFD modelling proves to be a cost-effective and time-efficient method for filter development in power generation applications.

References

- Abam, F. I., Effiom, S. O., & Ohunakin, O. S. (2016). *CFD evaluation of pressure drop across a 3-D filter housing for industrial gas turbine plants*. *Frontiers in Energy*, 10(2), 192–202.
- Effiom, Samuel & Ajour, James & Effiom, Precious-Chibuzo & Edem, Isuamfon & Ubi, Paschal & Abam, Fidelis & Diemuodeke, Ogheneruona. (2023). *Experimental study on the optimal performance of gas turbine (GT) inlet air filtration system for offshore application*. *Journal of Engineering and Applied Science*. 70. 10.1186/s44147-023-00303-8..
- Jin, Yunfeng & Liu, Chao & Tian, Xin & Huang, Haizhou & Deng, Gaofeng & Guan, Yunlong & Jiang, Dongxiang. (2021). *A novel integrated modeling approach for filter diagnosis in gas turbine air intake system*. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*. 236. 095765092110443. 10.1177/09576509211044392.
- Mohammed, A. S., Zaytoon, M. H. H., & Yiyun, L. X. (2021). Generalised models of flow of a fluid with pressure-dependent viscosity through porous channels: channel entry conditions. *International Journal of Physical Research*, 9(2), 84–91.
- Yi, J., Duan, J., Yuan, R., Bo, W., & Ruan, X. (2023). *Modeling and Cleaning Performance Optimization of Conical Filter Cartridge of Gas Turbine Intake Filter*. *Processes*, 11(9), 2584. <https://doi.org/10.3390/pr11092584>.