CFD Analysis And Evaluation Of Performance For Centrifugal Fan Using Wind Tunnel Testing

Mr.M.Dineshkumar[2], S.Manickavel[2], K.Paraneetharan[3], M.Rohith[4]
[1, 2, 3, 4] Dept of Mechanical Engineering
[1, 2, 3, 4] Kongunadu college of Engineering and Technology

Abstract- *This study investigates the performance of a centrifugal fan using a combination of Computational Fluid Dynamics (CFD) simulations and wind tunnel testing. The primary objective is to evaluate the fan's airflow characteristics, pressure rise, and overall efficiency under varying operational conditions. CFD simulations are utilized to model the fan geometry, including the impeller, casing, and diffuser, allowing for an in-depth analysis of flow patterns, pressure distribution, and velocity profiles. These simulations are performed with a focus on optimizing fan design for improved efficiency. Wind tunnel testing is then conducted to validate the CFD results by directly measuring key performance parameters such as airflow rate, static pressure, and fan power. The results from both methods are compared to assess the accuracy of the CFD model, and any discrepancies are used to refine the simulation. The study concludes that the integration of CFD analysis and wind tunnel testing provides a comprehensive approach to evaluating centrifugal fan performance, offering insights for design optimization and improved operational efficiency in practical applications.*

Keywords- Centrifugal fan, Computational Fluid Dynamics (CFD), Wind tunnel testing, Flow analysis, Pressure rise, Fan efficiency, Performance evaluation, Aerodynamic optimization, Flow rate, Turbulence modeling, Fan design, Experimental validation, Power consumption.

I. INTRODUCTION

Traditional methods of fan performance testing often rely on empirical data and physical wind tunnel experiments, which, while reliable, can be time-consuming, costly, and limited in scope. In recent years, Computational Fluid Dynamics (CFD) has become an increasingly popular tool in fan design and analysis. CFD allows for the simulation of airflow within the fan system, offering valuable insights into the behavior of air or gases at various stages of the fan’s operation, from the inlet to the outlet. This capability enables engineers to optimize fan geometry, predict performance under different operating conditions, and assess factors such as efficiency, noise, and pressure distribution.

This paper presents a combined approach to evaluate the performance of a centrifugal fan using both CFD simulations and wind tunnel testing. The study aims to assess the accuracy of CFD predictions and highlight the strengths and limitations of using numerical simulations for centrifugal fan analysis. By integrating both methods, the research seeks to develop a comprehensive understanding of centrifugal fan performance, enabling optimization of design, operational efficiency, and reliability in real-world applications.

II. COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems involving fluid flows. In the context of centrifugal fans, CFD is an invaluable tool for simulating and predicting the behavior of air or gas as it moves through the fan's components, such as the impeller, casing, and diffuser. The application of CFD in fan design and performance evaluation offers numerous advantages, including detailed insights into flow patterns, pressure distributions, and performance characteristics under varying conditions.

2.1 Benefits of CFD in Centrifugal Fan Analysis

- Design Optimization: CFD allows for iterative testing of fan designs without the need for physical prototypes. By adjusting parameters such as blade angle, geometry, and casing design, engineers can optimize fan performance for specific applications (e.g., improving efficiency, reducing noise, or achieving a specific pressure rise).
- Flow Visualization: CFD provides detailed visualization of airflow patterns inside the fan. This includes velocity vectors, streamlines, and turbulence structures, which can reveal potential flow issues, such as separation zones, dead spots, or areas of high shear stress that could negatively affect performance.
- Efficiency Prediction: CFD simulations help predict the fan's efficiency by calculating the total power required to drive the fan and comparing it to the power delivered to the airflow. Efficiency can be

optimized by fine-tuning fan geometry and operational parameters, ensuring the fan operates at peak performance.

- Cost and Time Savings: CFD analysis significantly reduces the time and costs associated with physical testing. Multiple design variations can be simulated quickly, allowing for rapid prototyping and reducing the need for extensive physical testing or rework.
- Scalability: CFD models can be scaled to represent different fan sizes and operating conditions, making it easier to apply the same principles to larger or smaller fans, or to predict how the fan will perform under different environmental conditions (e.g., temperature, humidity).

III. CENTRIFUGAL FAN

A centrifugal fan, also known as a centrifugal blower, is a type of mechanical fan commonly used in various industrial, commercial, and residential applications to move air or gases through a system. The fan works based on the principle of centrifugal force, which increases the velocity and pressure of the airflow as it moves through the fan's components.

The operation of a centrifugal fan is driven by the rotation of an impeller. The impeller consists of blades or vanes mounted on a shaft, and as the impeller spins, air or gas is drawn into the fan through the inlet (usually the center of the fan). The air is then directed radially outward (at a right angle to the intake direction) by the impeller blades. This centrifugal motion imparts kinetic energy to the air, increasing its velocity. The fast-moving air exits through the fan outlet, and the increased velocity leads to an increase in pressure.

- **Air Inlet (Suction Side):** The fan sucks in air at low pressure through the intake, which is usually located at the fan's center.
- **Impeller:** The rotating blades of the impeller accelerate the air radially outward.
- **Air Outlet (Discharge Side):** The accelerated air exits the fan at a higher pressure and velocity.

This process of converting the rotational kinetic energy of the impeller into the kinetic energy of the airflow is how centrifugal fans create airflow and pressure.

IV. WIND TUNNEL TESTING

Wind tunnel testing involves physically testing the centrifugal fan in a controlled environment, typically in a laboratory setup. A scale model or the actual fan can be tested in a wind tunnel to directly measure the performance and validate the CFD results.

4.1 Steps involved in Wind Tunnel Testing

Test Setup:

- The centrifugal fan is placed in the wind tunnel, and airflow is directed through the fan.
- The fan is usually mounted on a dynamometer to measure the mechanical power required to drive the fan.
- Pressure transducers and flow meters are used to measure static and total pressure, as well as airflow rate.

Measuring Performance Parameters

Pressure Rise: Measure the pressure difference across the fan (static and total pressure).

Flow Rate: Determine the airflow through the fan under different conditions (inlet velocity, fan speed).

Fan Efficiency: Calculate the efficiency using the input power (measured by the dynamometer) and the output airflow pressure.

Power Consumption: Measure the power required to operate the fan at different operating points.

Operating Points:

- Perform tests at various fan speeds, airflow rates, and static pressures to obtain a performance curve.
- Typical operating curves include the **fan curve** (pressure vs. flow) and the **power curve** (power vs. flow).

4.2 Benefits of Wind Tunnel Testing

- Provides real-world performance data and allows for validation of CFD predictions.
- Captures complex flow effects that may not be easily predicted by CFD alone, such as boundary layer separation and turbulence effects.
- Allows for the direct measurement of noise, vibrations, and other operational characteristics of the fan.

V. CFD AND WIND TUNNEL TESTING COMPARISON AND INTEGRATION

After conducting both CFD simulations and wind tunnel tests, it's essential to compare the results from both methods. The comparison can help validate the accuracy of the CFD predictions and highlight any discrepancies or limitations in the model. Here's how to approach the evaluation:

- **Validation:** Compare the pressure rise, flow rate, and fan power values obtained from CFD simulations with the measured data from wind tunnel testing.
- **Optimization:** Use CFD to optimize the fan design based on the insights gathered from wind tunnel tests.
- **Data Correlation:** Check for consistency between CFD-predicted performance curves and the wind tunnel experimental curves.
- **Error Analysis:** Identify areas where CFD might be overestimating or underestimating fan performance and refine the models accordingly.

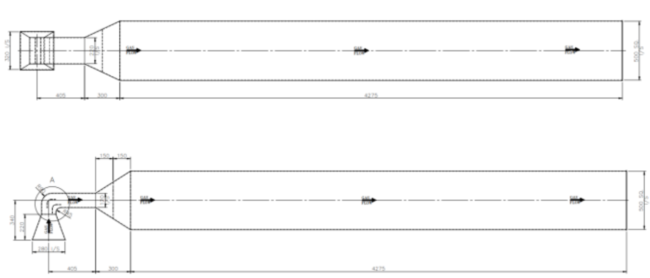
VI. EXPERIMENTAL SETUP

6.1Define the Objective of the Simulation

Before starting the CFD analysis, clearly define the objectives of the study. These may include:

- Performance evaluation of the centrifugal fan (e.g., airflow, pressure, efficiency).
- Validation of CFD results against experimental or wind tunnel data.
- Fan design optimization to improve performance or energy efficiency.

6.2 Geometry Creation



ALL DIMENSIONS ARE IN MM

6.3 Mesh Generation

- Mesh Refinement: Ensure that the mesh is refined around critical areas such as the fan blades, inlet, and outlet. These areas experience the highest gradients of velocity and pressure.
- Boundary Layer Mesh: Apply finer mesh near the fan surfaces to resolve the boundary layer and capture flow characteristics close to the walls (e.g., no-slip conditions).
- Volute Casing: Use a coarser mesh in the volute casing compared to the rotor and blade regions, as flow behavior is less complex there.

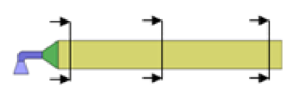
6.4 Boundary Condition

DESCRIPTION	VALUE	UNIT
Volume Flow rate	1	m^3/s
Density	1.204	Kg/m^3
Dynamic viscosity	1.8×10^{-5}	Pa*s

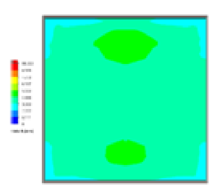
6.5 Run the Simulation

- After setting all boundary conditions, initialize the solver and begin the simulation.
- Depending on the complexity of the model and mesh, this process may take minutes to hours

7. Results and Discussion



VELOCITY PLOT @ D-D



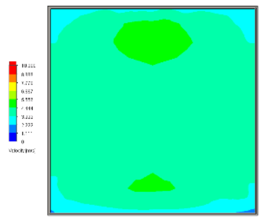
%RMS=4.33%

This image represents a velocity contour plot from a Computational Fluid Dynamics (CFD) simulation.

- The color legend on the left represents velocity magnitude (in m/s), ranging from 0 m/s (blue) to 10 m/s (red).
- The contour plot shows how velocity is distributed across the domain.
- The shape of the domain suggests an enclosed flow system, possibly a ventilated room, duct, or chamber.

- The dominant cyan color (~5-6 m/s) suggests that the majority of the flow field has a moderate velocity.
- If this is a ventilation system, the green zones might indicate poor air circulation (potential dead zones).
- If this is a duct flow, these variations could be caused by obstructions or flow separation.
- If turbulence models were used, the lower velocity regions could indicate vortex formation or recirculation effects.

VELOCITY PLOT @ E-E

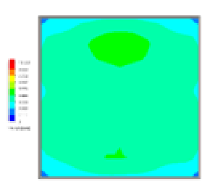


%RMS=3.69%

This image represents a velocity contour plot from a Computational Fluid Dynamics (CFD) simulation. Below is a detailed analysis of the results:

- The color legend on the left represents velocity magnitude (m/s), ranging from:
  0 m/s (blue) → Low velocity regions
  10 m/s (red) → High velocity regions
- The image appears to show a 2D flow field inside an enclosed space (possibly a room, duct, or chamber).
- The dominant cyan color (~5-6 m/s) suggests that much of the flow field maintains a moderate velocity.
- The majority of the flow field is cyan, indicating moderate velocity values (~5-6 m/s) and these areas indicate a velocity drop (~4 m/s).

VELOCITY PLOT @ F-F

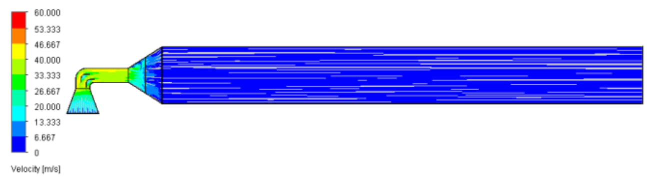


%RMS=3.20%

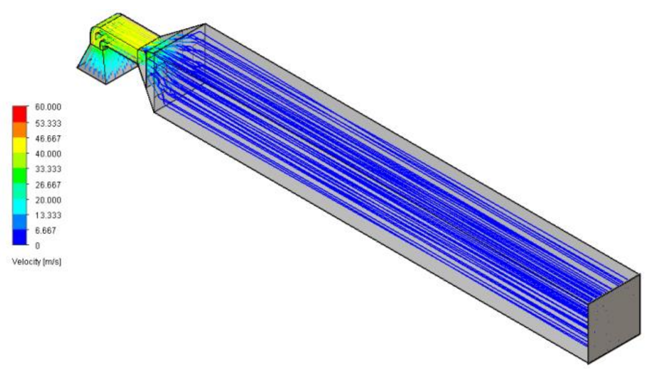
This image represents a velocity contour plot from a Computational Fluid Dynamics (CFD) simulation. Below is a detailed description and analysis of the results.

- The color legend on the left indicates the velocity magnitude in meters per second (m/s).
- The velocity range spans from:
  0 m/s (dark blue) → Lowest velocity regions.
  10 m/s (red) → Highest velocity regions.
- The plot appears to represent airflow distribution in an enclosed space, likely a room, chamber, or duct system.
- The majority of the domain is cyan (~5-6 m/s), indicating a moderate velocity zone.
- A green region at the top center suggests a velocity drop (~4 m/s), possibly caused by flow stagnation or a recirculation zone.
- A smaller green region near the bottom center also indicates a localized velocity reduction and the dark blue areas near the corners indicate low-velocity regions (0-2 m/s)

7.1 . FLOW TRAJECTORIES



ELEVATION



ISOMETRIC VIEW

This image represents a velocity contour plot from a Computational Fluid Dynamics (CFD) simulation, likely analyzing flow through a nozzle, duct, or wind tunnel system. The color-coded scale on the left denotes the velocity magnitude in meters per second (m/s), ranging from:

0 m/s (dark blue) – lowest velocity regions
60 m/s (red) – highest velocity regions

- The flow enters through a funnel-like nozzle or intake, where the velocity is relatively low (blue to green regions, around 6-20 m/s).
- As the flow contracts and moves through the curved pipe, the velocity increases (green-yellow transition),

indicating acceleration due to the narrowing of the passage.

VIII. CONCLUSION

Combining CFD analysis with wind tunnel testing provides a comprehensive approach to evaluating centrifugal fan performance. CFD allows for detailed, high-resolution simulations of flow behavior, while wind tunnel testing provides real-world, measurable performance data. Together, these methods help design more efficient fans, predict performance under different conditions, and optimize fan operation for specific applications.This integrated approach leads to improved product design, reduced prototype testing time, and a better understanding of how the fan will perform in actual operational environments.

Appendix

A. Fan Specifications and Geometry

1. Fan Model Specifications
   - Fan Type: Backward Curved Centrifugal Fan
   - Impeller Diameter: 300 mm
   - Impeller Blades: 6 blades, backward-curved design
   - Motor Power: 1.5 kW
   - Fan Speed: 2800 RPM
   - Inlet Diameter: 150 mm
   - Outlet Diameter: 180 mm
   - Housing Type: Cylindrical casing with a volute design
   - Material: Aluminum for the impeller and mild steel for casing

2. Fan Performance Data (Tested and Simulated)
   - Table showing airflow, pressure, and power consumption for various operating points in both CFD and experimental wind tunnel testing.

B. CFD Simulation Setup

1. Geometry Modeling
   - Detailed steps on how the 3D model of the centrifugal fan was created in CAD software (e.g., SolidWorks or CATIA).
   - Mention of any design modifications or simplifications made for simulation purposes.

2. Meshing
   - Mesh Type: Structured mesh with refinement near critical areas such as the impeller blades, inlet, and outlet.
   - Mesh Quality: The number of cells, skewness, orthogonality, and aspect ratio.
   - Element Type: Hexahedral elements were used in the mesh for higher accuracy in flow predictions.

3. Boundary Conditions
   - Inlet: Mass flow rate of 2 kg/s.
   - Outlet: Pressure outlet condition, with atmospheric pressure as the reference.
   - Walls: No-slip boundary condition applied to the fan blades, casing, and inlet/outlet walls.

4. Turbulence Model:
   - Model Used: k-ε turbulence model with standard wall functions for near-wall flow.
   - Solver Type: Steady-state, pressure-based solver.

5. Solver Settings:
   - Convergence Criteria: Residuals below 1E-5 for velocity, pressure, and turbulence parameters.
   - Time Step: N/A (steady-state simulation).

C. Wind Tunnel Testing Setup

1. Test Rig
   - Wind Tunnel Size: 2m x 2m open-circuit wind tunnel.
   - Fan Mounting: Fan mounted on a motorized test stand with adjustable speed control.

Measurement Instruments

- Pressure Transducers: Used to measure static and total pressures at the fan inlet and outlet.
- Flow Meter: Rotameter used to measure the volumetric flow rate.
- Dynamometer: To measure power consumption of the fan motor.
- Temperature and Humidity Sensors: Monitored to account for environmental conditions during testing.

Future Scope

- Advanced Turbulence Models and Unsteady Simulations
- Optimization of Fan Design
- Hybrid CFD and Experimental Methods
- Noise Prediction and Control

- Sustainable and Eco-Friendly Design

Acknowledgement

We express our sincere gratitude to our guide, Mr.M.Dineshkumar.,M.E., for their invaluable guidance, support, and encouragement throughout this research onCFD Analysis and Evaluation of Performance for Centrifugal fan using Wind tunnel testing. Their insights and expertise have been instrumental in shaping this study.

We would also like to thank **Kongunadu college of Engineering & Technology, Department of Mechanical Engineering** for providing the necessary resources and facilities. Special appreciation goes to my colleagues and family for their continuous motivation and support.

Finally, We acknowledge all the researchers whose work has contributed to the foundation of this study.

S.MANICKAVEL, K.PARANEETHARAN, M.ROHITH

Kongunadu college of Engineering & Technology

REFERENCES

[1] Li, Y., Zhang, Q., & Wang, L.,'Innovations in ESP optimization for gas- liquid ducts: Advanced electrode materials, computational fluid dynamics, and real- time monitoring. Journal of 42 the Environmental in a Engineering'., Vol 140, No.8, pp.60-81, April 2023

[2] J. Park, H. Kim, S. Lee, "A Study on the Performance Characteristics of Centrifugal Fans Using CFD" in Journal of Mechanical Engineering Science, vol. 227, pp. 2324–2332, June 2013.

[3] M. A. A. Sharif, R. A. I. Abidin, K. A. Hassan, "Performance Analysis and Optimization of a Centrifugal Fan Using CFD" in Energy, vol. 76,pp. 96-105, April 2014.

[4] F. C. Di Maio, A. N. Tavares, A. R. M. A. Lima, "CFD Simulation of a Centrifugal Fan: A Comparative Study with Experimental Measurements" in Applied Thermal Engineering, vol. 150, pp. 734–745, May 2019