ENGINEERING SIMULATION

The Engineering of Fluid Dynamics: A Comprehensive Technical Guide to Bladeless Technology and CFD Simulation in Ansys

Published: November 3, 2025 | Index ID: #53

The evolution of fluid machinery has historically been defined by the optimization of blades—rotary components designed to transfer energy between a fluid and a mechanical system. However, a significant paradigm shift has emerged in the last decade, popularized by the consumer electronics industry and now expanding into aerospace, propulsion, and renewable energy: **bladeless technology**. From the iconic Dyson Air Multiplier to Mohyi Labs' bladeless propulsion systems and cutting-edge wind turbines, the removal of visible rotating components offers advantages in safety, noise reduction, and aerodynamic efficiency. At the heart of these innovations lies **Computational Fluid Dynamics (CFD)**, specifically through the use of industry-standard tools like **Ansys Fluent** and **Ansys Mechanical**.

Designing a system that moves air or generates thrust without external blades requires an intricate understanding of complex physics, including the **Coanda Effect**, **Bernoulli's Principle**, and **fluid entrainment**. This article provides a high-level technical analysis of the engineering principles behind bladeless systems, the methodology for simulating these phenomena in Ansys, and the broader implications for the future of mechanical design.

The Theoretical Framework: Physics of Bladeless Fluid Movement

To the casual observer, a bladeless fan seems to defy physics. However, the mechanism is a sophisticated application of fluid mechanics. The internal architecture of a bladeless device typically contains a high-pressure impeller hidden within the base. This impeller drives air into a hollow ring, or airfoil-shaped frame. The following physical principles govern the resulting airflow:

1. The Coanda Effect

The **Coanda Effect**is the tendency of a fluid jet to stay attached to a convex surface. In bladeless fans, air is forced out through a narrow slit (often between 1.0mm and 1.6mm wide) that runs along the inner circumference of the ring. As the air exits this slit, it follows the curved profile of the ring's inner surface. This creates a localized area of low pressure, which is crucial for the next stage of fluid movement.

2. Fluid Entrainment and Inducement

Bladeless technology functions as a **flow multiplier**. The primary air (the air pushed by the hidden impeller) is only a fraction of the total output. As the primary air moves across the surface of the ring, it creates a vacuum-like effect that pulls in the surrounding air. This process is known as **entrainment**. Furthermore, the air behind the fan is drawn through the center of the ring due to the pressure differential, a process called **inducement**. In high-performance designs, the ratio of total air output to primary air intake can be as high as 15:1 or 20:1.

3. Bernoulli’s Principle

The shape of the ring mimics an aircraft wing (an airfoil). According to **Bernoulli's Principle**, as the velocity of the air increases along the curved surface, its static pressure decreases. This pressure gradient is the primary driver for the massive air movement observed in devices like the Dyson Air Multiplier.

Technical Analysis: Simulating Bladeless Systems in Ansys Fluent

Designing these systems requires precise control over geometry and airflow. **Ansys Fluent** is the premier tool for modeling these interactions, allowing engineers to visualize pressure gradients and velocity vectors that are impossible to observe with the naked eye. The following workflow outlines the standard procedure for a **Parametric Study of a Bladeless Fan Geometry**.

Phase I: Geometry Preparation and SpaceClaim Modeling

The first challenge in simulating a bladeless system is the **aspect ratio** of the geometry. The exit slit is extremely narrow compared to the overall diameter of the fan. Engineers use **Ansys SpaceClaim** or **DesignModeler** to create a fluid domain that encompasses both the internal high-pressure plenum and the external atmospheric environment. Precision in the curvature of the "Coanda surface" is vital, as even a micron-level deviation can lead to flow separation, where the air detaches from the surface and causes turbulence.

Phase II: Advanced Meshing Strategies

The mesh is the foundation of a reliable CFD simulation. For bladeless fans, a **hybrid meshing approach** is often used:

- Inflation Layers: To accurately capture the Coanda effect, high-density inflation layers are placed along the walls of the airfoil ring. This ensures that the Boundary Layer physics (the velocity gradient near the wall) are resolved.
- Refinement Zones: A high-density mesh is required at the slit exit where the highest velocity gradients occur.
- Poly-Hexcore Mesh: Modern versions of Ansys Fluent (2024 and beyond) utilize Poly-Hexcore meshing to reduce cell count while maintaining accuracy in the far-field fluid domain.

Phase III: Turbulence Modeling and Solver Setup

Because the flow in bladeless systems is inherently turbulent and involves high-velocity jets, selecting the right turbulence model is critical. The **Shear Stress Transport (SST) k-omega model** is the industry standard for these applications because it excels at predicting flow separation under adverse pressure gradients. Engineers must also decide between **Steady-State** (for average performance metrics) and **Transient** simulations (to analyze noise generation and vortex shedding).

Metric	Laminar Model	k-epsilon Model	SST k-omega Model
Near-Wall Accuracy	Low	Medium	High
Flow Separation	Poor	Poor	Excellent
Computational Cost	Low	Medium	Medium-High
Ideal Use Case	Low-speed pipe flow	General industrial mixing	Aerodynamic surfaces / Bladeless fans

Case Study: Mohyi Labs and Bladeless Propulsion

Beyond cooling fans, the **Ansys Startup Program** has empowered companies like **Mohyi Labs**to rethink aerial propulsion. Traditional propellers pose safety risks and are susceptible to damage in cluttered environments. Mohyi Labs' bladeless propulsion system leverages the same Coanda-based air multiplication principles to generate lift for drones.

By using **Ansys Fluent**, Mohyi Labs was able to optimize the "thrust-to-power" ratio by iteratively changing the airfoil thickness and the angle of the air jet. This digital prototyping reduced the need for expensive physical wind tunnel testing and allowed for the development of a propulsion system that is not only safer but also significantly quieter than traditional multi-rotor configurations.

Mathematical Modeling of Flow Multipliers

The performance of a bladeless fan is often quantified by the **Magnification Ratio (M)**. This is defined as:

M = Q_total / Q_primary

Where:

- Q_total is the total volume flow rate exiting the ring.
- Q_primary is the volume flow rate of the air pushed into the plenum by the internal impeller.

Using the **Navier-Stokes equations**, Ansys Fluent solves for the conservation of mass and momentum across the fluid domain. In a bladeless system, the **entrainment velocity (U_e)**can be estimated using the relationship between the jet velocity and the surrounding air density. Engineers aim to maximize U_e while minimizing the energy consumption of the internal motor.

Structural Integrity and Aeroacoustics in Ansys Mechanical

Bladeless doesn't mean no moving parts; it means no *exposed* moving parts. The internal impeller spins at high RPMs, creating vibration and heat. **Ansys Mechanical** is used to perform **Fatigue Analysis** on the impeller and the housing. Furthermore, the high-velocity air exiting the slit can generate high-frequency noise. Through **Aeroacoustic Simulation**, engineers can identify the specific geometric features causing "whistling" or "humming" and modify the design to ensure a silent operation.

Comparison: Traditional Bladed Fans vs. Bladeless Technology

Feature	Traditional Bladed Fan	Bladeless (Air Multiplier) Technology
Safety	High risk (exposed rotating blades)	Highest (no moving parts exposed)
Maintenance	Difficult (dust accumulates on blades)	Easy (simple wipe-down of the ring)
Airflow Quality	Buffeting (choppy air)	Constant, laminar-like stream
Energy Efficiency	High at low speeds	Highly efficient due to entrainment ratios
Noise Profile	Mechanical vibration + blade chop	Aeroacoustic wind noise (tunable)

Practical Implementation: Design Optimization Workflow

For engineers looking to implement bladeless technology in industrial or consumer products, the following **Step-by-Step Optimization Guide** is recommended:

1. Define Performance Targets

Establish the required **Cubic Feet per Minute (CFM)**and the maximum allowable power consumption. This will dictate the size of the internal motor and the diameter of the multiplier ring.

2. Conduct a Parametric Study

Use **Ansys DesignXplorer** to run a series of simulations varying the following parameters:

- Slit Width: Affects the velocity of the primary jet.
- Airfoil Profile: Determines the effectiveness of the Coanda effect.
- Plenum Volume: Ensures uniform air distribution around the entire ring circumference.

3. Analyze Pressure Distribution

Inspect the **Static Pressure Contours**in Ansys Fluent. A successful design will show a distinct low-pressure zone on the inner surface of the ring, confirming that entrainment is occurring.

4. Validation with Experimental Data

Compare simulation results with **Anemometer**readings from a physical prototype. If the simulated velocity profiles match the physical measurements within a 5% margin, the CFD model is considered validated.

NASA and the Future of Bladeless Simulation

The significance of this technology is further validated by **NASA's** interest. NASA has awarded contracts for advanced modeling and simulation capabilities to Ansys, focusing on the next generation of aircraft. These include concepts for **Distributed Electric Propulsion (DEP)**where bladeless or semi-bladeless systems could be integrated into the wings of aircraft to improve lift-to-drag ratios and reduce the carbon footprint of aviation.

As we look toward **2025 and 2026**, the integration of **Artificial Intelligence (AI)** within the Ansys ecosystem—such as **Ansys SimAI**—will allow for even faster optimization of bladeless geometries. Instead of waiting hours for a CFD solve, AI models trained on previous simulation data will be able to predict the performance of a new bladeless design in seconds.

The Broader Engineering Implications

The transition from bladed to bladeless technology represents more than just a change in aesthetics; it is a fundamental shift in how we manipulate the fluid environment. By leveraging the Coanda effect and fluid entrainment, engineers can create systems that are safer, quieter, and more reliable. The reliability stems from the reduction of mechanical complexity; fewer exposed moving parts mean fewer points of failure and lower maintenance requirements over the lifecycle of the product.

The role of simulation tools like **Ansys Fluent** cannot be overstated. Without the ability to model the invisible interactions of air at high velocities, the development of the Dyson fan or Mohyi Labs' propulsion system would have required decades of trial-and-error. Today, through **High-Performance Computing (HPC)** and advanced turbulence modeling, we can refine these designs to their theoretical limits. As bladeless technology moves into larger-scale applications, such as wind energy and industrial HVAC systems, the synergy between fluid dynamics theory and computational power will remain the primary driver of innovation. The future of fluid motion is not found in the rotation of a blade, but in the precise, simulated control of the air itself.

Baca Juga (Artikel Terkait)

- **Comprehensive Guide to Finite Element Analysis (FEA) for Free Fall Drop Testing: Engineering Principles and Applications**
  URL: http://alghafgolden.com/finite-element-analysis-drop-test-simulation-guide.pdf
- **The Comprehensive Guide to Abaqus Documentation: Technical Architecture, FEA Methodologies, and Implementation Strategies**
  URL: http://alghafgolden.com/comprehensive-guide-abaqus-documentation-technical-fea-strategies.pdf
- **Mastering Autodesk Nastran In-CAD 2015: A Technical Deep Dive into FEA Integration and Structural Analysis**
  URL: http://alghafgolden.com/mastering-autodesk-nastran-in-cad-2015-technical-guide.pdf
- **The Comprehensive Guide to Autodesk Nastran In-CAD 2017: Technical Implementation, FEA Principles, and Engineering Strategy**
  URL: http://alghafgolden.com/comprehensive-guide-autodesk-nastran-in-cad-2017-fea.pdf

Tags: #Ansys Fluent #CFD Simulation #Bladeless Technology #Coanda Effect #Fluid Dynamics