Contra-Rotating Axial Fan Design Concept for Leaf Blowers

By Christopher Sands and Ralph Peter Mueller, CFturbo, Inc. Brooklyn, NY January 2024

1. Motivation and Objectives

It seems leaf blowers are everywhere in our neighborhoods, very often disturbingly noisy. There is a need for excellent aerodynamic and aeroacoustics design for such devices.

Three leaf blower configurations were designed using CFturbo: a single-stage design, a dual-stage design, and a contra-rotating design. The goal of the design study is to minimize the rotational speed required to meet the performance targets and minimize the swirl downstream of the nozzle. The design point volume flow rate was set to 500 cfm at 25,000 rpm with a total pressure difference of 3,000 Pascals.

Although the relationship between sound output and rotational speed is not directly proportional, higher rotational speeds often result in remarkably increased noise levels due to higher airflow velocity, higher blade tip speeds, higher frequency noise components, and increased flow instabilities. A robust acoustic design optimization would require extensive simulation and testing. By reducing the downstream swirl, the user has improved directional control over the airflow leaving the device.

2. Conceptual Design

A single-stage, a dual-stage, and a contra-rotating axial fan design was created to showcase all the potential configurations a user could build in CFturbo for this type of device.

The **single-stage** fan design includes an entry nozzle, the stage—an impeller followed by stationary guide vanes, a diffuser, and an outlet pipe. **Figure A** shows the meridional view of the single-stage axial fan configuration.

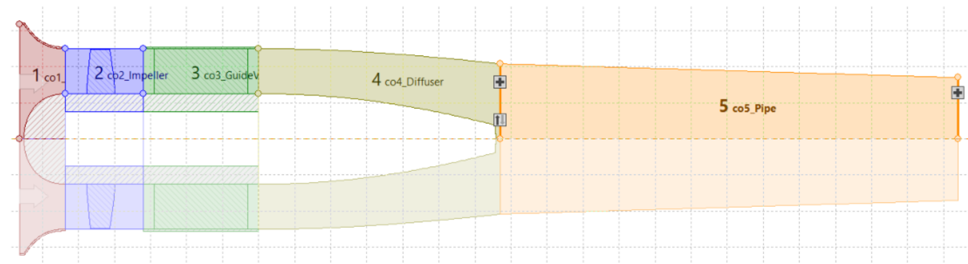


Figure A: Single-Stage Fan

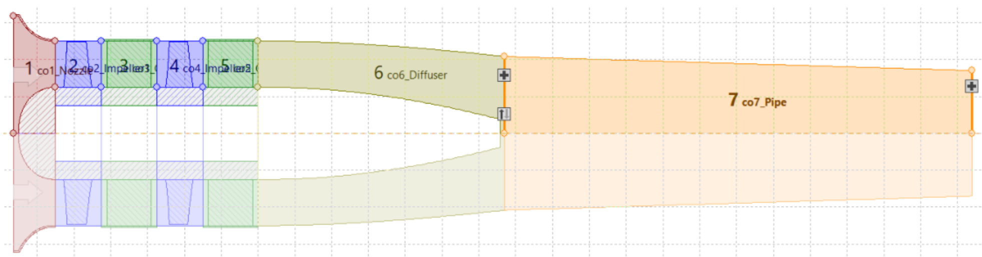


Figure B: Dual-Stage Fan

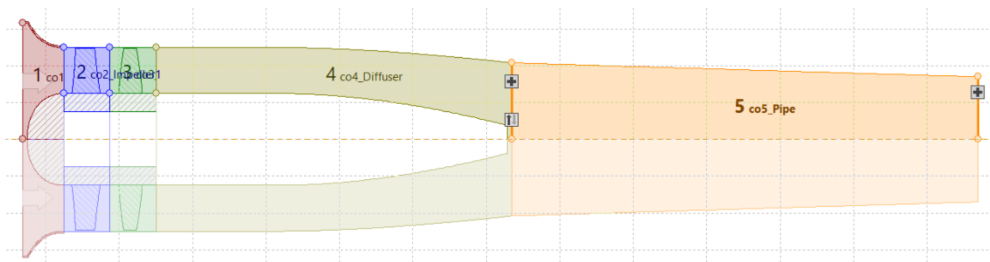


Figure C: Contra-Rot Fan

The **dual-stage** fan design includes an entry nozzle, the stage—two consecutive pairs of impellers and stationary guide vanes, a diffuser, and an outlet pipe. **Figure B** shows the meridional view of the dual-stage axial fan configuration.

The **contra-rotating** fan design includes an entry nozzle, the stage—two consecutive impellers rotating in opposite directions, a diffuser, and an outlet pipe. **Figure C** shows the meridional view of the contra-rotating axial fan configuration.

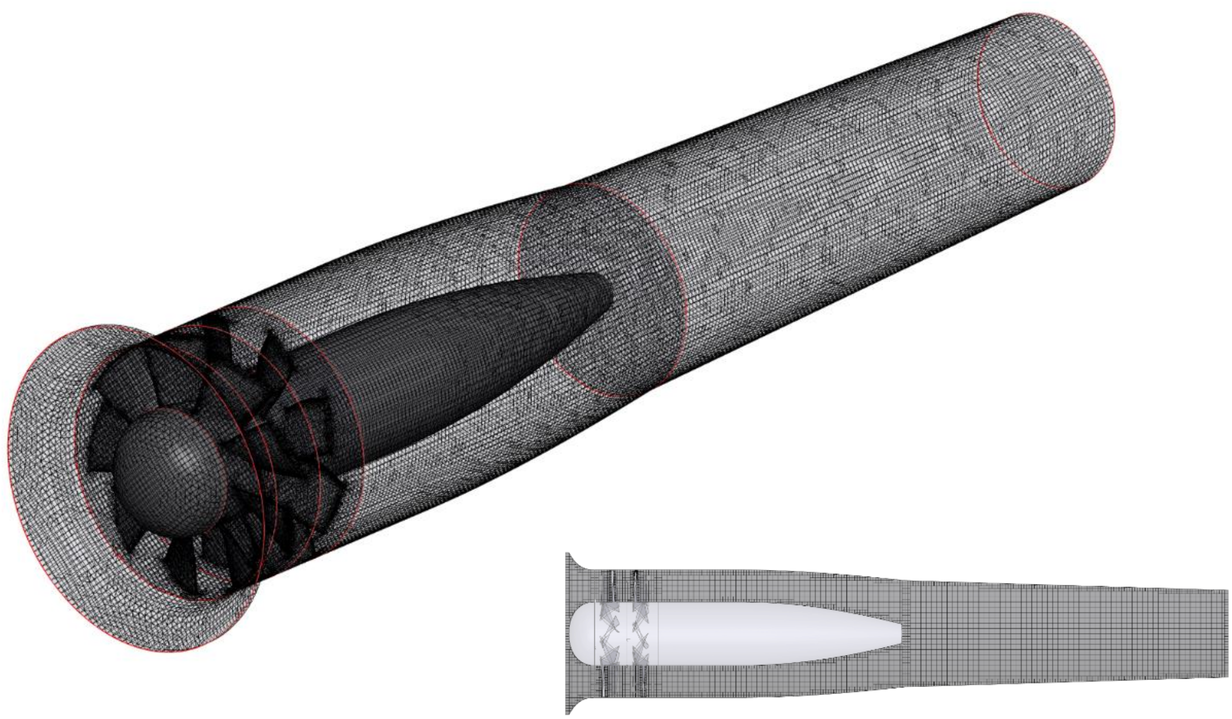
All three design configurations had an inlet diameter of 5 inches and a total axial length of 20.5 inches.

3. Flow Simulation:

A hexahedral, binary-tree mesh with approximately 2 million nodes was created using SimericsMP. **Figure D** shows the mesh of the contra-rotating axial fan design. For each simulation, a total pressure boundary condition was applied at the inlet of the nozzle, and a volumetric flowrate boundary condition was applied at the outlet of the pipe. The rotating components within the device were given a rotating wall boundary condition with a specified rotational speed. Both the outlet volumetric flow rate and rotational speed were parameterized for sweeping.

At a set outlet volumetric flow rate and rotational speed, steady-state simulation was performed, and that solution was used to initialize a subsequent transient simulation. The simulation process is automated. The steady-state simulations applied an MFR (Multiple Frame of Reference) approach. The transient simulations worked with moving mesh technology. The transient simulations utilized a total of 360 timesteps or 3 degrees per timestep (= 120 timesteps per one full revolution), a second-order upwind scheme for the velocity calculation, and a first-order upwind scheme for the pressure calculation.

Figure D: Contra-Rotating Axial Fan, Simerics MP Mesh



4. Key Results:

Data regarding the **aerodynamic performance of the fan** was extracted from CFturbo SMP (Simerics MP) and plotted below. Each leaf blower configuration was initially designed for a rotational speed of 25,000 rpm. However, with the preliminary design, the dual-stage and contra-rotating axial fan designs over-performed at the design point rotational speed. A second series of simulations was performed on the dual-stage and contra-rotating axial fan designs to determine which operating rotational speed results in a total pressure difference of approximately 3,000 Pascals; **Figure E** shows these results.

Figure E: Total Pressure Difference vs. Volume Flow Rate

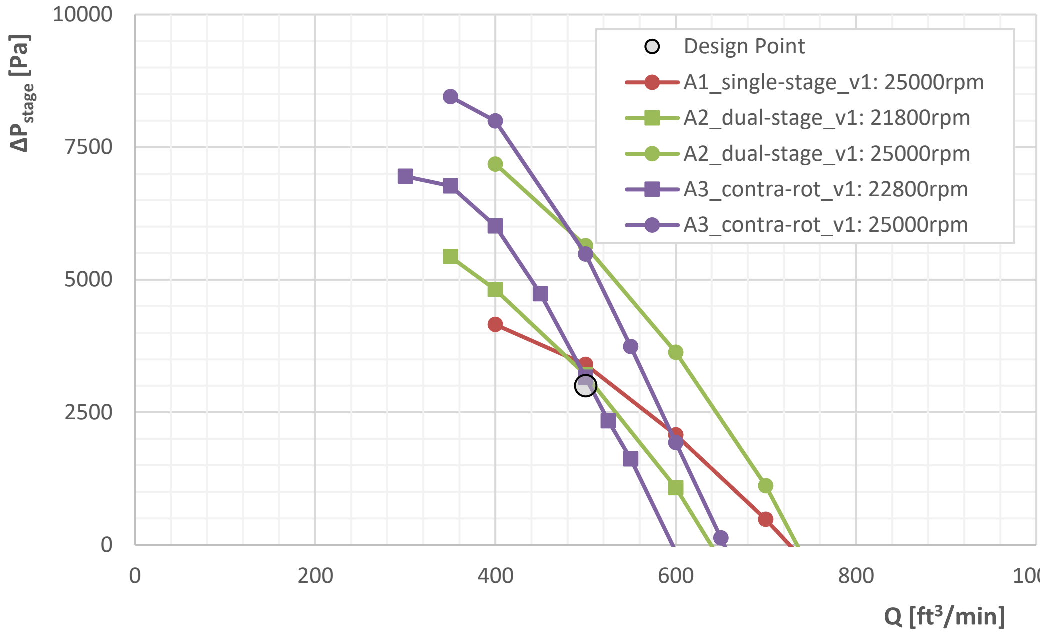
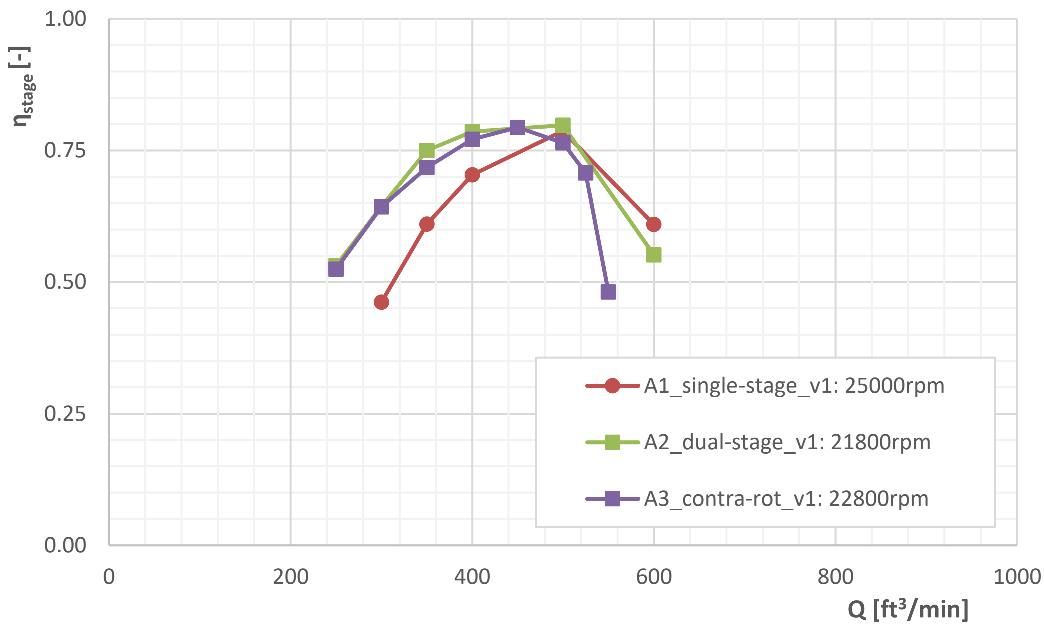


Figure F: Total-to-Total Efficiency vs. Volume Flow Rate



Comparing the **total-to-total efficiency** of each fan concept, the contra-rotating model displayed similar efficiency values to the dual-stage fan design, especially at lower flow rates. At the design point, all three designs displayed aerodynamic stage efficiencies peak higher than 75%. This behavior in efficiency is seen in **Figure F.** Because the contra-rotating axial fan had comparable results in performance and efficiency at the nominal design speed, a more in-depth simulation study was conducted on the design; a complete performance map was created using rotational speeds of 20,000 to 30,000 rpm.

Total pressure difference stage, exit nozzle velocity, total-to-total stage efficiency, impeller efficiency, and shaft power are displayed below. The specific design of the flow path will determine the pressure losses and, thus, the system curve of the leaf blower. An aerodynamically excellent design leads to low total pressure losses and enables high exit velocities. A high aerodynamic efficiency over a wide speed range minimizes power consumption.

Figure G: Contra-Rotating Axial Fan, Total Pressure Difference, Stage

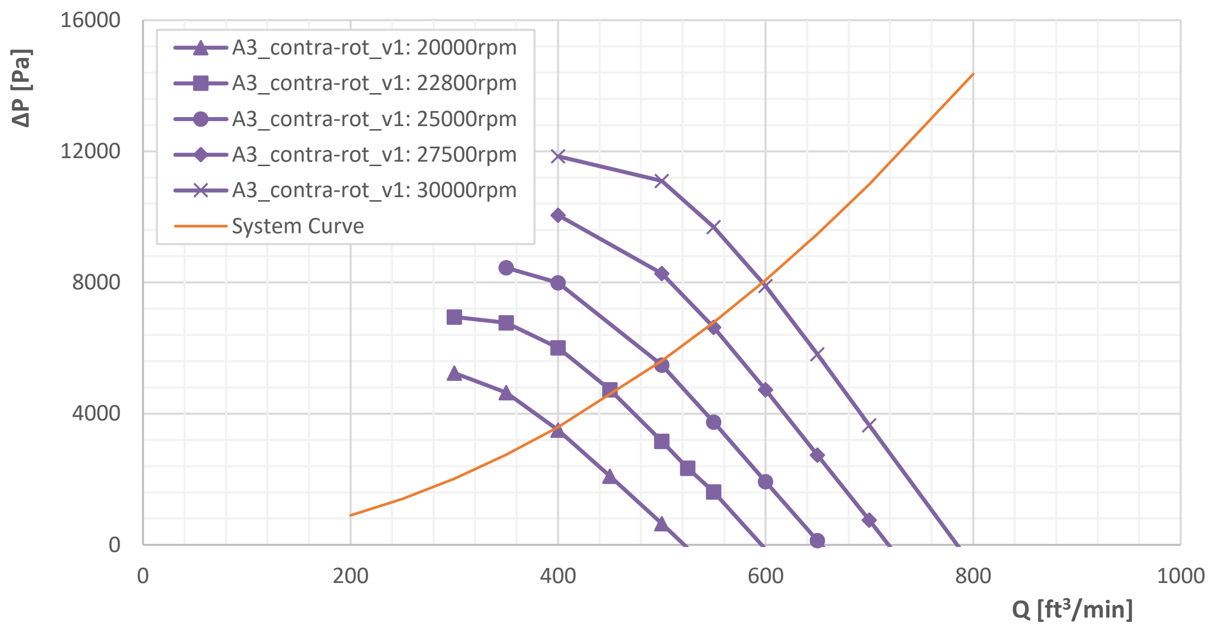
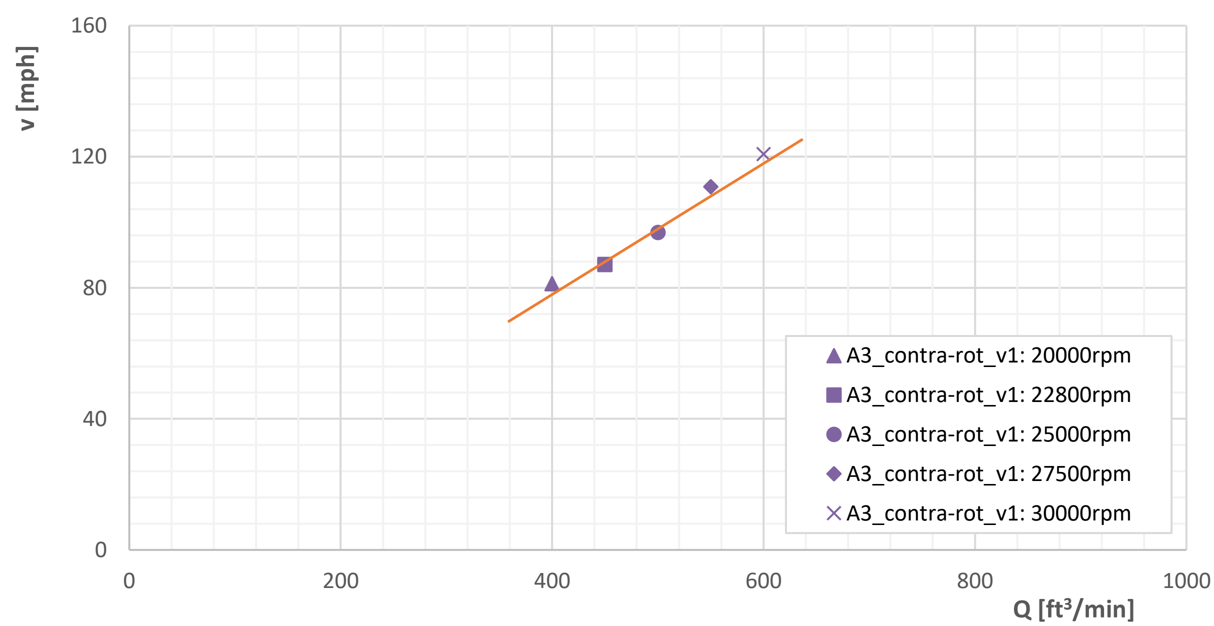


Figure H: Exit Nozzle Velocity vs. RPM vs. Flow Rate



The exit nozzle velocity can be adjusted by changing the rotational speed. An exit nozzle with a variable end cross-sectional area would affect both the exit velocity and the system curve.

Figure I: Contra-rotating Axial Fan, Total-to-total Efficiency, Stage (Impeller1 + Impeller2)

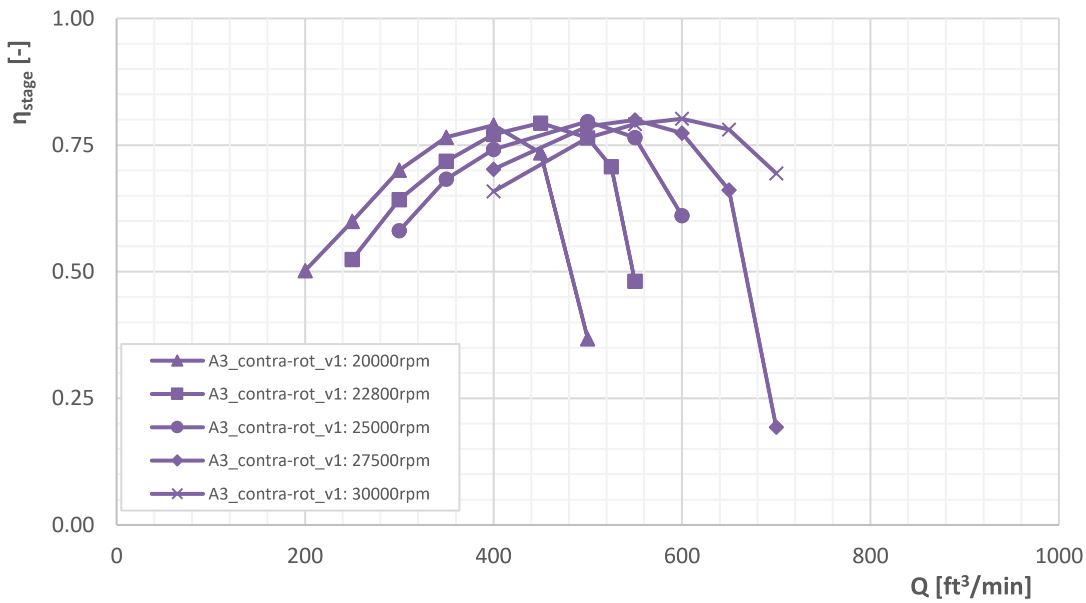


Figure J: Contra-rotating Axial Fan, Total-to-total Efficiency Comparison (Impeller1, Impeller2)

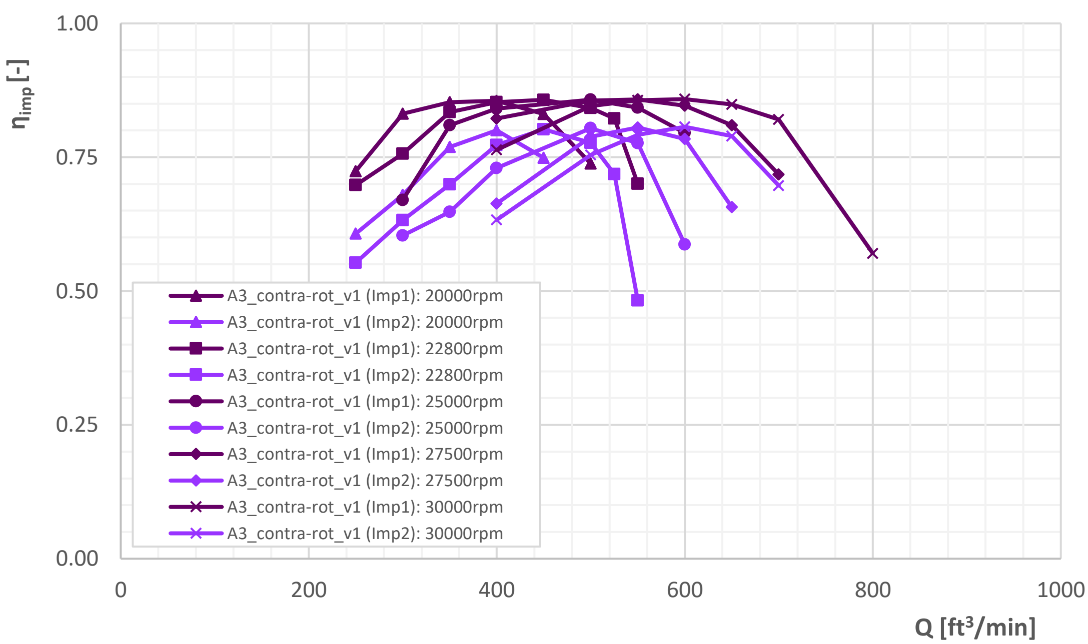
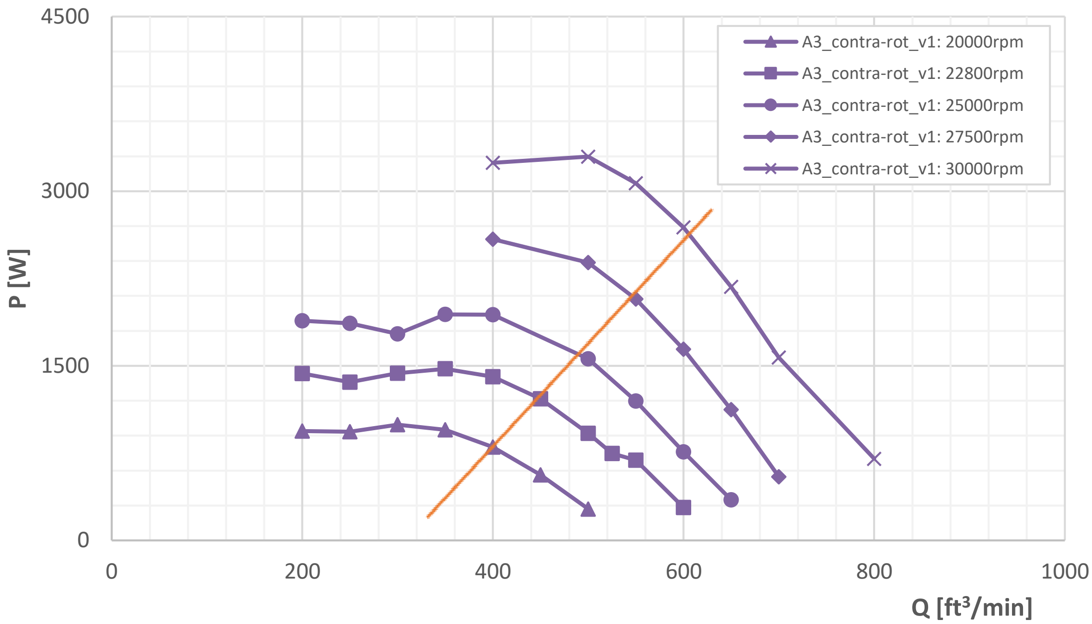


Figure K: Contra-rotating Axial Fan, Shaft Power



5. Conclusions

Based on the results of transient CFD simulation, a contra-rotating axial fan design can achieve the target aerodynamic performance with comparable efficiency at lower revolutions per minute. As seen below in **Figure L**, the contra-rotating design significantly reduces the swirl downstream of the stage when compared to the single-stage axial fan design, resulting in lower system losses and a better controllable outflow pattern/exit nozzle velocity. Overall, aerodynamics and aeroacoustics have unique development potential, especially when considering the variable speed options of two individually controlled rotors.

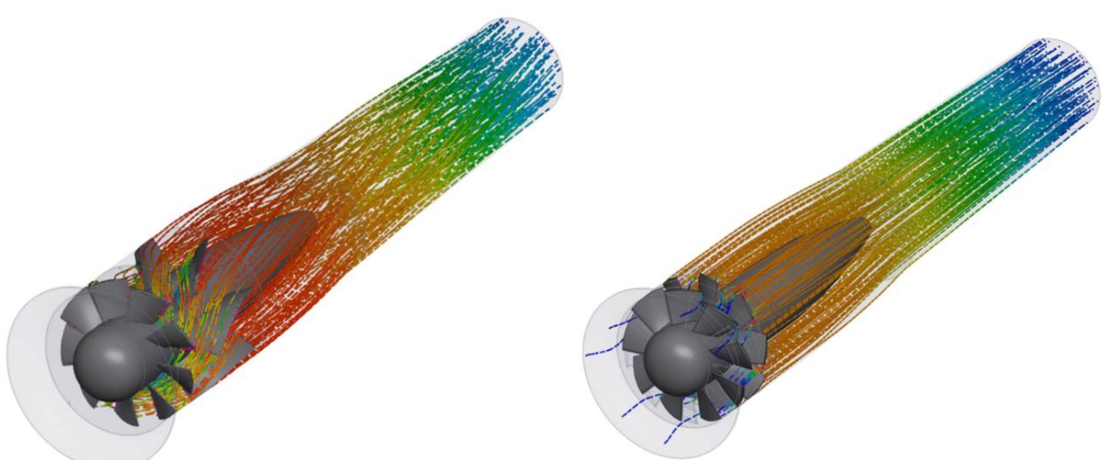


Figure L: Swirl Visualization of Single-stage Axial Fan (Left) at 25,000 RPM/500 CFM and Contra-rotating Axial Fan (Right) at 22,800 RPM/500 CFM

This can be quantitatively displayed with the magnitude of vorticity on both the exit of the nozzle and the exit of the outlet pipe, seen here in Figures M and N. Vorticity is a vector quantity that describes the local rotation of fluid particles within a fluid and is mathematically defined as the curl of the velocity vector field of the fluid, seen by Equation 1. High vorticity, seen in the single-stage axial impeller design, implies a strong tendency for rotation, while low vorticity suggests less flow rotation.

$$\boldsymbol{\omega} = \boldsymbol{\nabla} \times \vec{\mathbf{v}} \tag{1}$$

Figure M: Vorticity Magnitude of Single-Stage Axial Fan at 25,000RPM/500 CFM

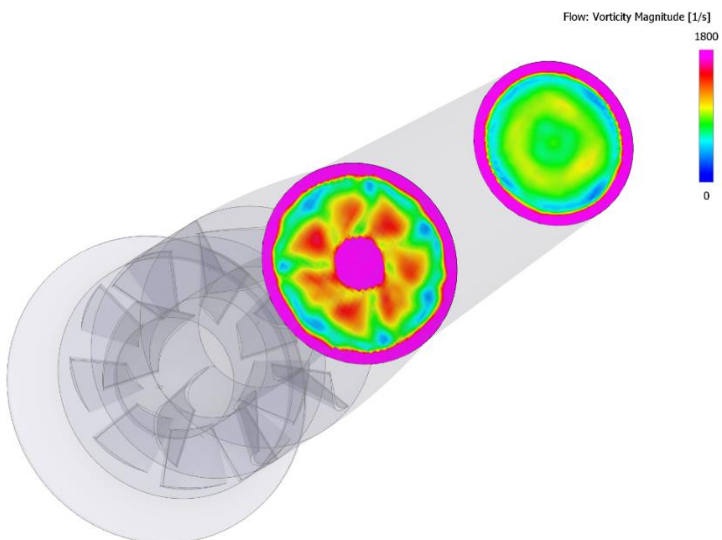
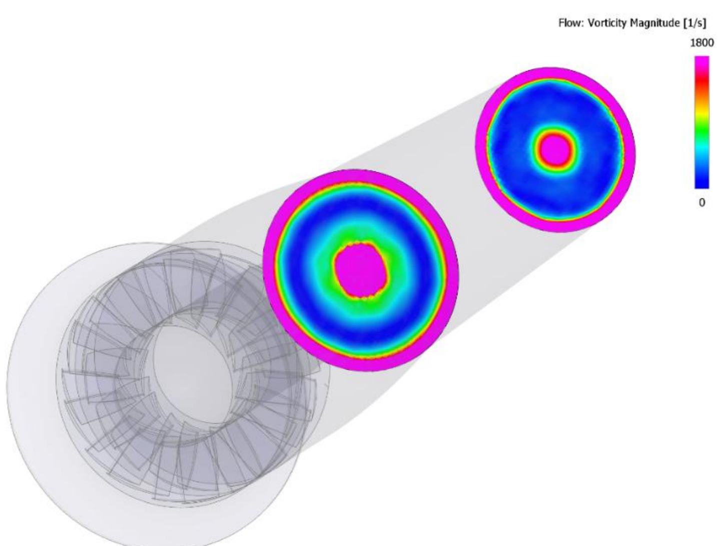


Figure N: Vorticity Magnitude of Contra-Rotating Axial Fan at 22,800RPM/500 CFM



About CFturbo

CFturbo (est. 2008) is headquartered in Dresden, Germany, with a major office in New York City, New York. The company is supported by a global network of distributors and has gained worldwide respect within the Turbomachinery community over the last ten years. CFturbo is dedicated to Turbomachinery design and related engineering services in designing rotating machinery and solving fluid flow and heat transfer problems. Our conceptual design software is the most user-friendly system available on the market—through its unrivaled, intuitive, and user-friendly design process, CFturbo software empowers every user, regardless of experience. The software can be used to design various turbomachinery-related devices, including pumps, fans, blowers, compressors, turbines, stators, and volutes. CFturbo, Inc. offers a variety of Turbomachinery engineering services, including aerodynamic and hydraulic designs, CFD and FEA simulation, rotating machinery optimization, mechanical design, prototyping, and testing. For more information, visit cfturbo.com.