

CFD Analysis of an Axial Fan Using ANSYS Fluent

Prepared by: Mohamed Aldreamly

Date: 13/2/2026

LinkedIn User Name: Mohammed AL Dreamly

www.linkedin.com/in/mohammed-al-dreamly-924317286

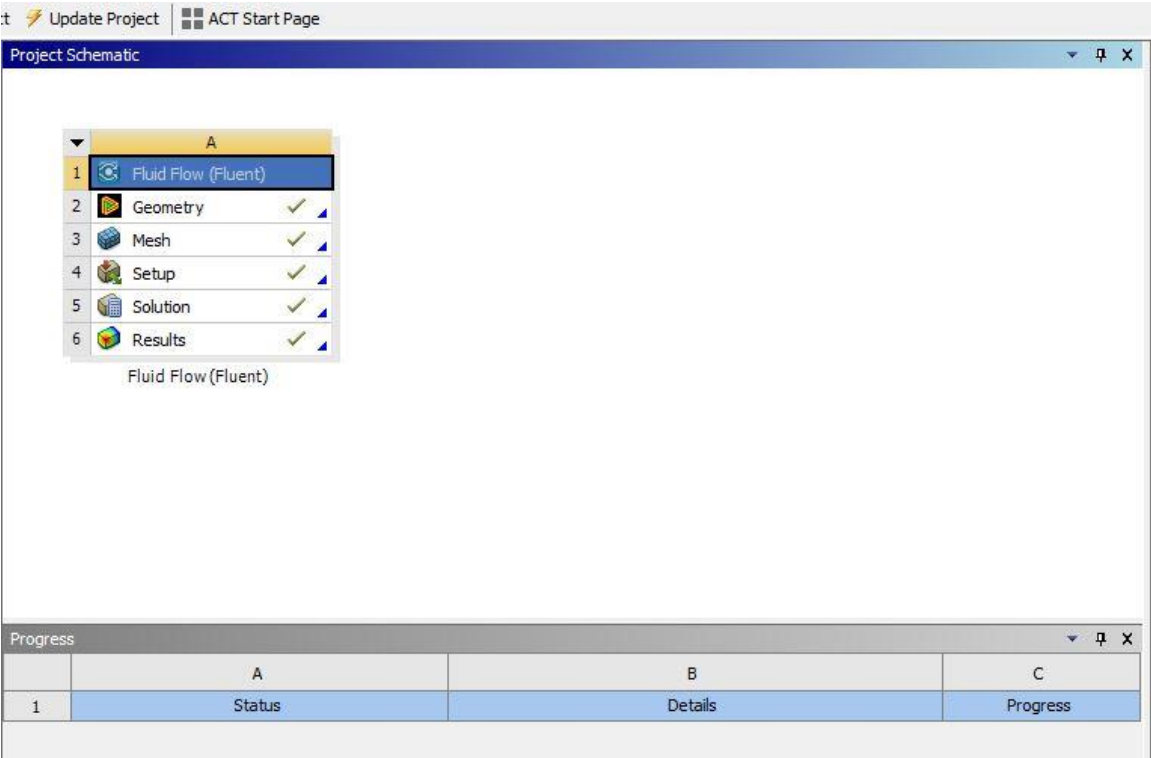
GitHub: <https://github.com/MohamedAldreamly>

1. Introduction

This project presents a Computational Fluid Dynamics (CFD) analysis of an axial fan operating inside an enclosed domain. The objective of the study is to investigate:

- Velocity distribution
- Pressure behavior
- Flow pattern (streamlines)
- Swirl and recirculation zones
- Overall aerodynamic performance

The simulation was performed using ANSYS Fluent (2025 R2 Student Version).

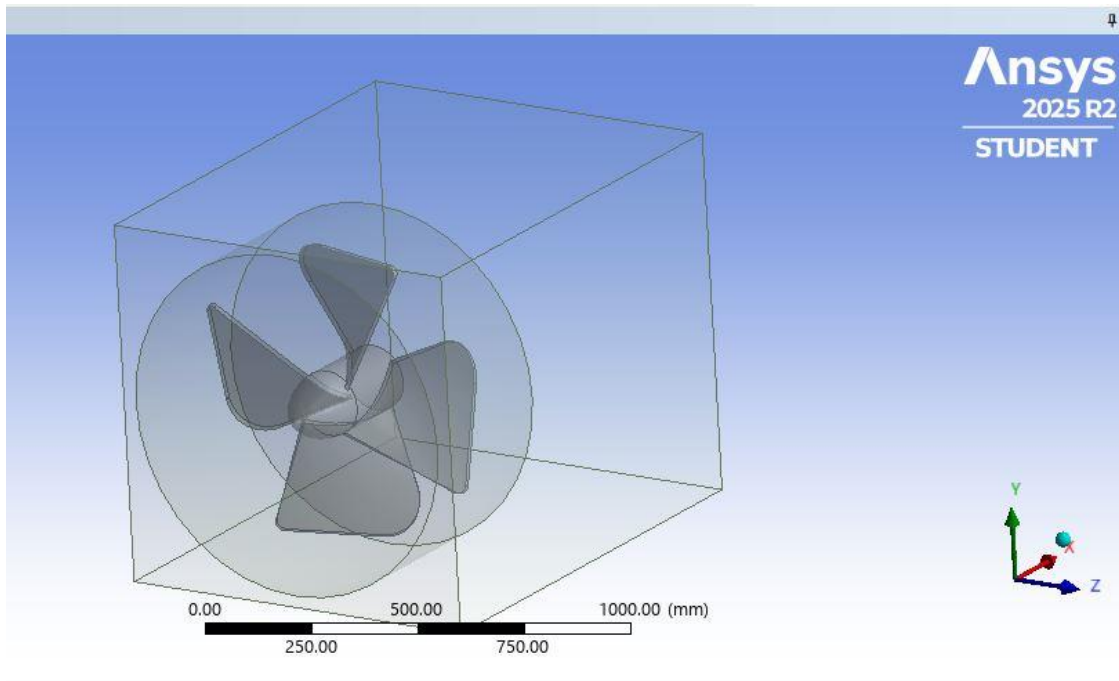


2. Geometry Description

The computational domain consists of:

- A 3-blade axial fan (solid body)
- A cylindrical rotating region (fluid zone)
- A surrounding stationary enclosure (fluid zone)

The rotating region was created around the fan to simulate rotational effects using the Sliding Mesh approach.



3. Computational Domain Setup

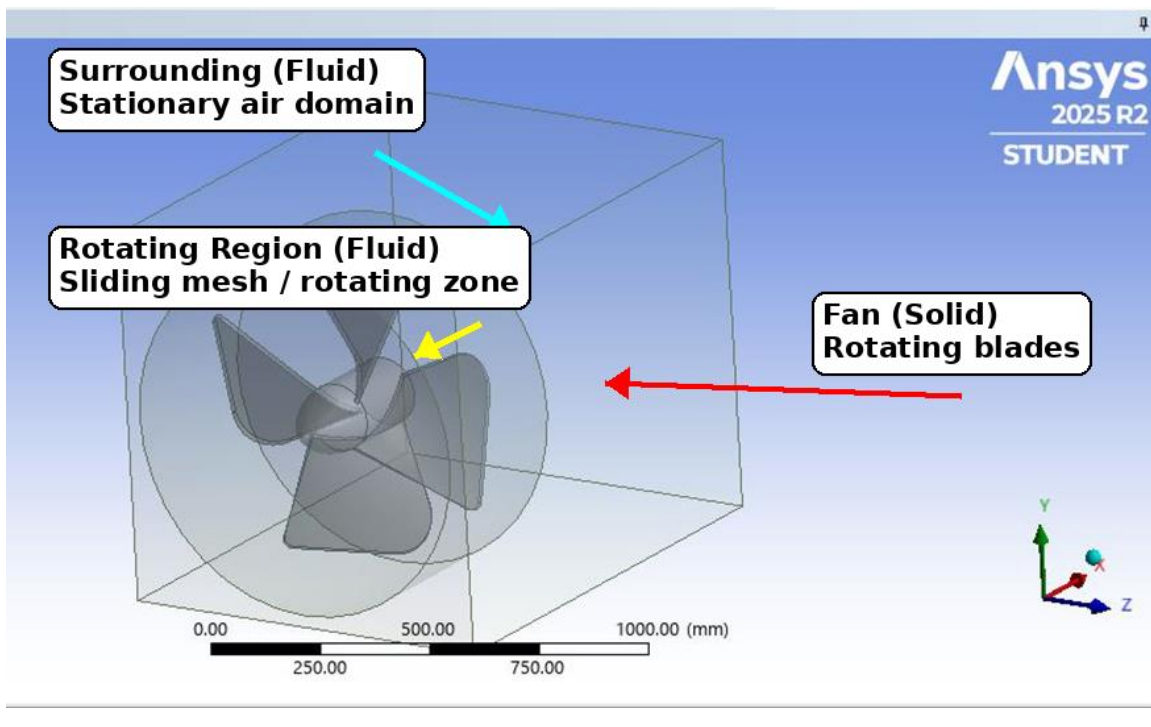
3.1 Domain Type

- 3D simulation
- External flow inside enclosure
- Air as working fluid

3.2 Regions

Region	Type	Description
Fan	Solid	Rotating blades
Rotating Region	Fluid	Sliding mesh zone
Surrounding	Fluid	Stationary air domain

The rotating region is defined to accurately capture the rotational effects of the fan blades, including centrifugal and Coriolis forces. The surrounding region represents the stationary air domain where the flow develops, spreads, and forms recirculation zones



4. Mesh Description

The computational mesh was generated using the **MultiZone method** in ANSYS Meshing to ensure structured and high-quality elements suitable for rotating flow simulations.

Meshing Method

- **Meshing Method:** MultiZone
 - **Decomposition Type:** Program Controlled
 - **Mapped/Swept Type:** Hexahedral
 - **Free Mesh Type:** Hexa Core
 - **Surface Mesh Method:** Program Controlled
 - **Sheet Body Method:** Quad Dominant
 - **Sweepable Body Method:** Sweep
 - **Element Order:** Global Setting
-

5. Physical Models

- **Time Formulation:** Transient
 - **Dynamic Mesh:** Enabled
 - **Mesh Motion Type:** Sliding Mesh
 - **Reference Frame:** Absolute
 - **Gravity:** Off
-

Pressure–Velocity Coupling

- Scheme: SIMPLE
 - Flux Type: Rhie–Chow
-

Spatial Discretization

- Pressure: Second Order
 - Momentum: Second Order Upwind
 - Turbulence: First Order (initially), then Second Order
-

Time Settings

Turbulence Model

- Reynolds Stress Model (as seen in residuals)

Parameter	Value
Rotational Speed	500 RPM
Time Step Size	0.001 s
Number of Time Steps	500
Iterations per Step	20

Time step size was selected to represent approximately 3° of rotation per step.

6. Boundary Conditions

Boundary	Type	Value
Inlet	Pressure Inlet	0 Pa
Outlet	Pressure Outlet	0 Pa
Walls	No-slip	
Rotating Region	Sliding Mesh (500 RPM)	

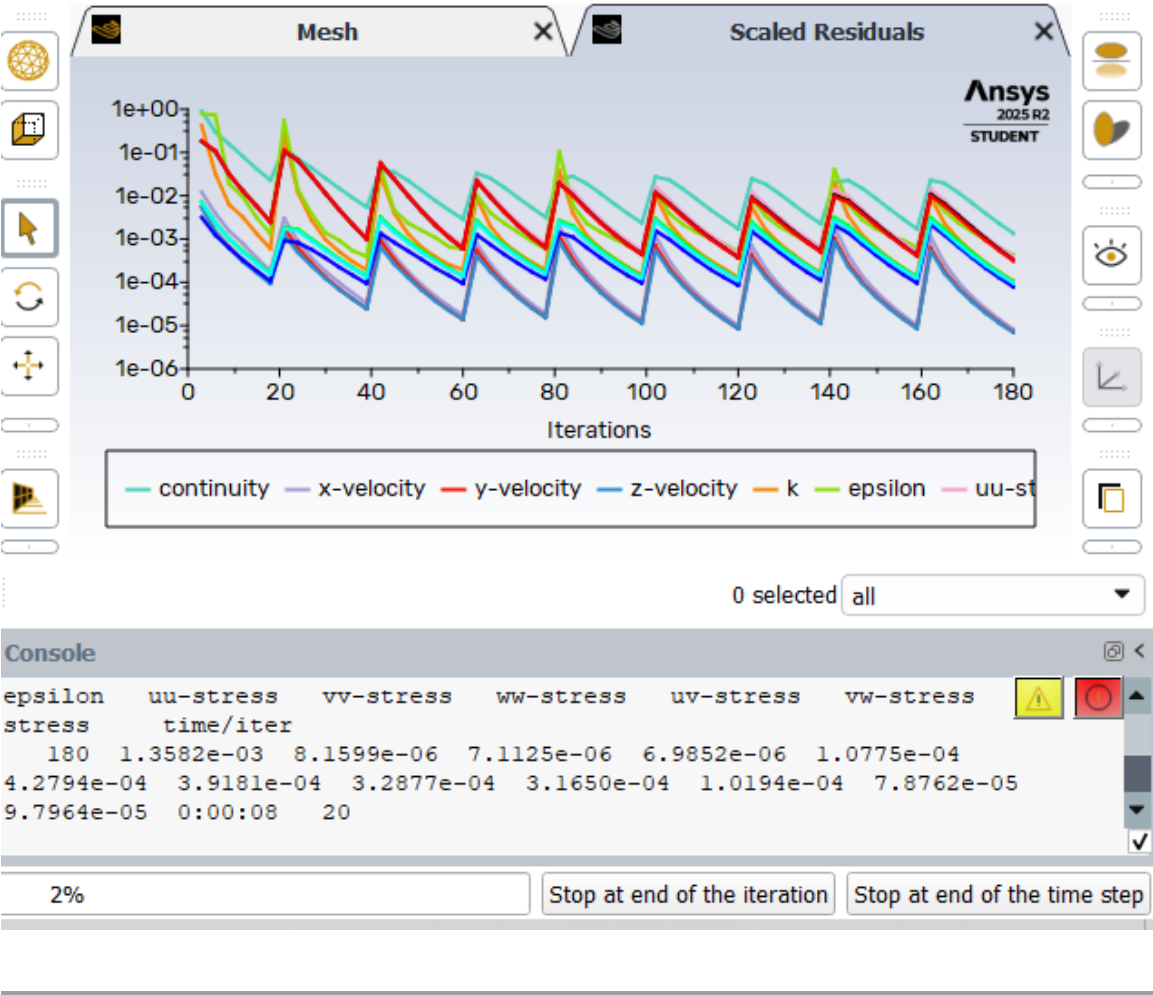
Turbulence intensity at inlet: 5%

Viscosity ratio: 10

7. Results and Discussion

Residual Convergence

Scaled residuals exhibit a periodic saw-tooth behavior, which is characteristic of transient sliding-mesh simulations. Residuals decrease significantly within each time step, indicating successful inner-iteration convergence, while the repeated spikes correspond to mesh motion and the updated flux exchange across the rotating–stationary interface. The nearly constant peak-to-valley pattern confirms that the solution has reached a stable periodic regime associated with blade passage, which is an acceptable convergence indicator for rotating machinery simulations.



Velocity Distribution

The velocity distribution was analyzed using contour plots and streamline visualization in the stationary reference frame. The maximum velocity recorded in the domain reaches approximately 46–48 m/s, primarily located near the blade tips and immediately downstream of the fan.

High-Velocity Regions

The highest velocities occur at the outer radius of the blades (tip region). This behavior is physically expected due to the relation: $V_{\text{tip}} = \omega r$ where the linear velocity increases with radius. Since the tangential speed is proportional to the radial distance from the axis of rotation, the blade tips transfer the highest momentum to the surrounding air. This confirms that the fan is effectively accelerating the flow in the intended direction.

A high-velocity jet is clearly formed downstream of the fan, indicating successful momentum transfer from the rotating blades to the fluid.

Low-Velocity Regions

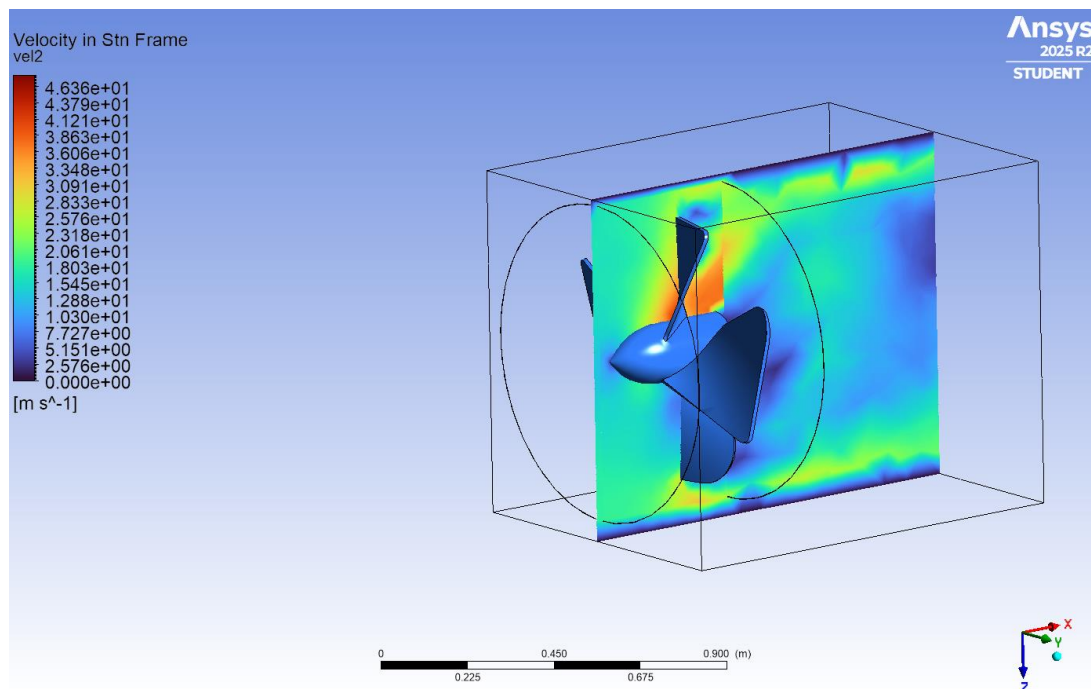
Low velocity zones are observed:

Near the hub region, due to smaller radial distance.

Close to the walls of the enclosure, due to the no-slip boundary condition.

In downstream recirculation zones where the flow loses momentum.

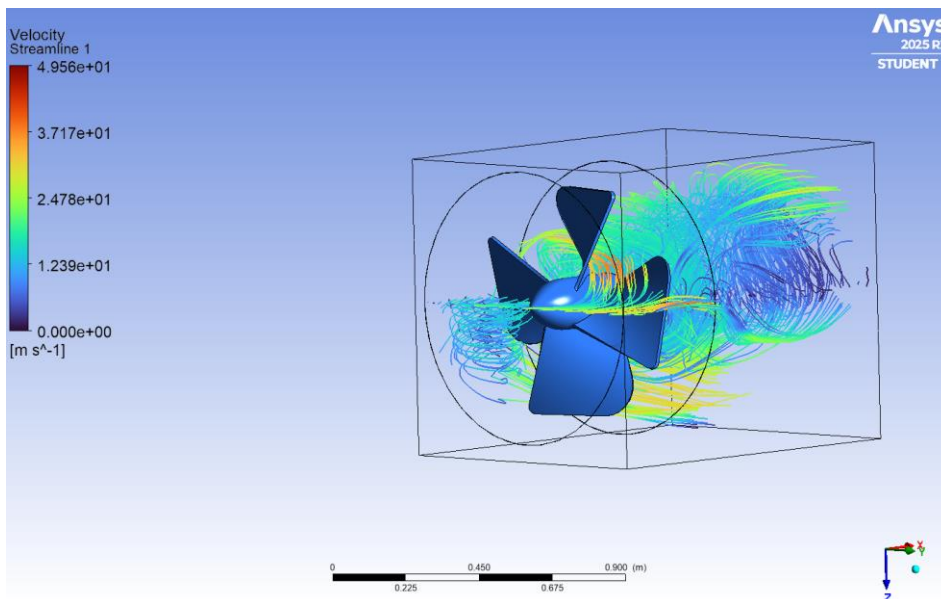
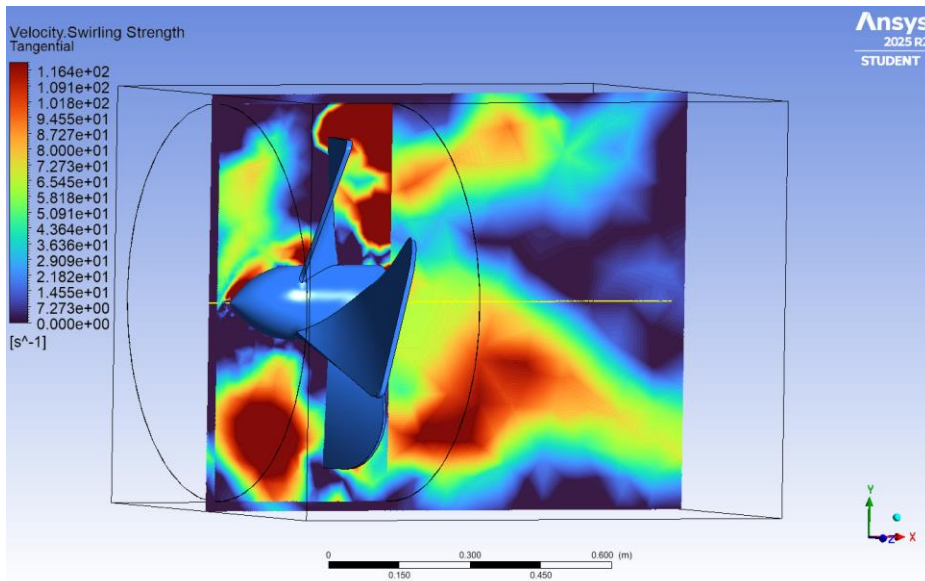
These regions indicate areas of reduced kinetic energy and possible local energy dissipation.



Swirl and Recirculation

The contour of swirling strength highlights the development of intense vortical structures downstream of the rotor. High swirl intensity regions are clearly concentrated near the blade tips and within the wake zone, indicating strong rotational motion induced by the blades. These vortical structures persist into the downstream region, confirming significant angular momentum transfer from the rotor to the flow.

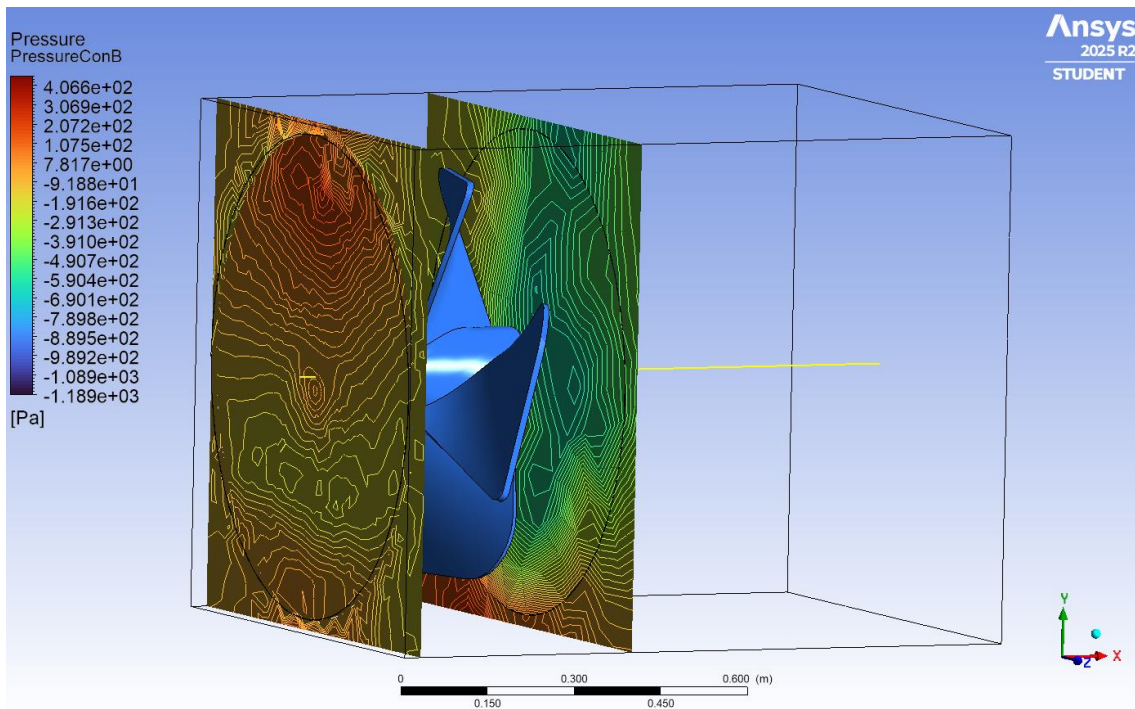
The distribution also reveals asymmetry in the wake, suggesting non-uniform flow mixing and localized recirculation zones within the enclosure. Such behavior is typical in confined rotating flows, where blade-induced vortices interact with enclosure boundaries and contribute to complex pressure and velocity gradients



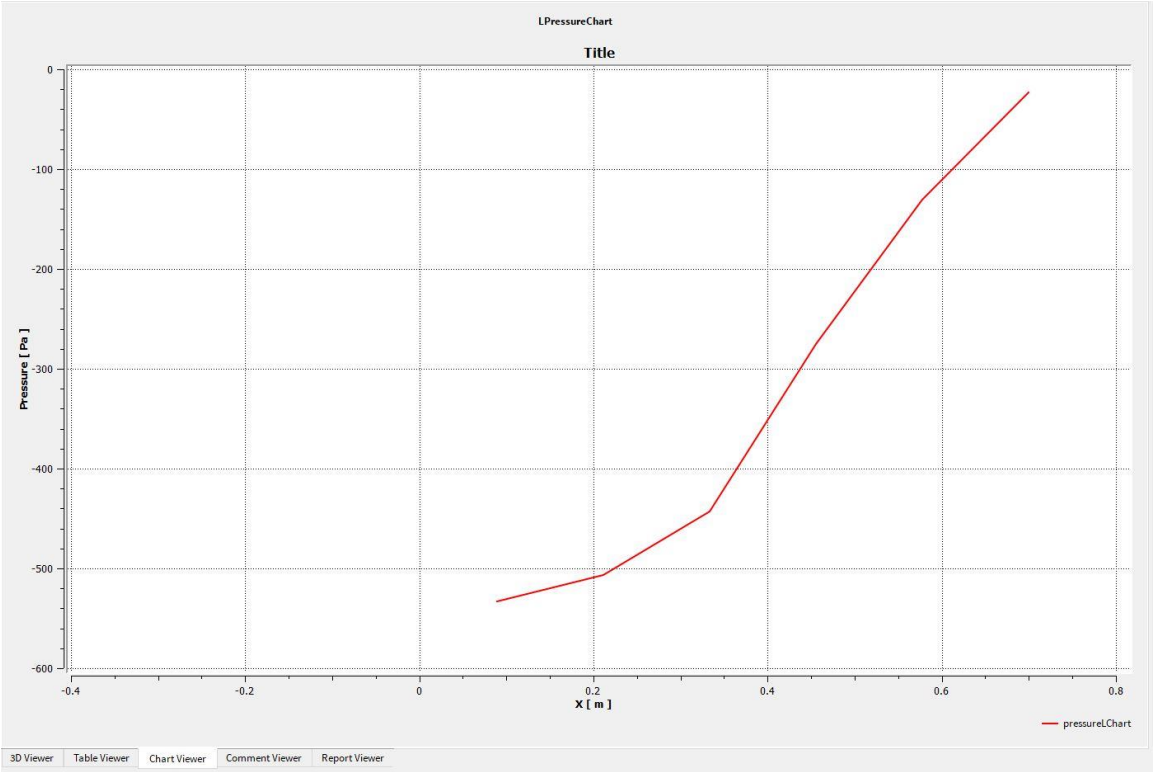
Pressure Behavior

Static pressure was evaluated using two reference planes located upstream and downstream of the rotor. The area-averaged values were $P_{\text{before}} = -86.99 \text{ Pa}$ and $P_{\text{after}} = -364.61 \text{ Pa}$, resulting in a static pressure variation of $\Delta P = -277.62 \text{ Pa}$ across the rotor region.

The negative pressure change indicates a significant downstream pressure reduction within the confined enclosure. This behavior is physically consistent with rotor-induced flow acceleration, where part of the mechanical energy imparted by the blades is converted into kinetic energy rather than static pressure rise. As the flow accelerates through the rotor plane, static pressure locally decreases according to Bernoulli's principle, while strong wake formation and swirling motion contribute to additional pressure losses.



The pressure distribution along the streamwise direction (X-axis), shown in Figure (X), further clarifies this mechanism. The pressure profile exhibits a steep gradient in the immediate vicinity of the rotor disk, identifying the region of strongest momentum exchange between the blades and the fluid. Downstream of the rotor, the pressure gradually recovers, although it remains lower than the upstream value due to wake mixing and confined-domain recirculation effects. These results confirm that within the enclosed configuration, the rotor primarily redistributes energy into rotational and axial kinetic components, generating a pronounced wake rather than producing a uniform static pressure rise.



9. Engineering Interpretation

The fan successfully generates:

- Axial flow
- High velocity jet
- Rotational swirl

However, strong internal recirculation indicates that:

- The enclosure size influences performance
 - Flow confinement increases turbulence
 - Some mechanical energy converts into vortical structures instead of useful thrust
-

10. Limitations

- Student version mesh limitation
 - No experimental validation
 - Domain confinement effect
 - Possible mesh refinement improvement near blade tips
-

11. Conclusion

The CFD analysis demonstrates that the axial fan operating at 500 RPM produces significant velocity acceleration and swirl motion within the enclosure.

Sliding mesh simulation captured transient blade interaction and periodic flow behavior. The results indicate realistic aerodynamic performance with expected vortex structures and jet formation.

The study confirms the importance of transient simulation for rotating machinery analysis.
