Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 7-8, 2026

CFD Analysis of Aerodynamic Forces on an S-CO₂ Compressor Blade at Various Incident Angles

Byung Chul Kim [a], Jeong Min Baek [a], Jeong Ik Lee [a*]
[a]*Dept. of Nuclear & Quantum Eng., KAIST,* 291 Daehak-ro, Yuseong-Gu, Daejeon 34141, Republic of Korea
[a*]*Corresponding author: jeongiklee@kaist.ac.kr*

***Keywords : S-CO2 Brayton cycle, Compressor blade, CFD, Aerodynamic characteristics, Incident angle**

1. Introduction

The supercritical carbon dioxide (S-CO_2) Brayton cycle is emerging as a next-generation power conversion system, offering a superior thermal efficiency and a compact system footprint in the turbine inlet temperature range of 450 to 650 ˚C. These advantages primarily stem from the significant reduction in compression work when the fluid operates near the CO_2 critical point (31.1 ˚C, 7.38 MPa) [1]. While the overall efficiency of the cycle is heavily dependent on compressor performance, the aerodynamic characteristics near the critical point—where physical properties change non-linearly—remain insufficiently understood [2].

Historically, the development of turbomachinery has been built upon a vast amount of design experience and data based on air. To leverage this existing knowledge for S-CO_2 turbomachinery, it is essential to define the aerodynamic similarity between air and S-CO_2. Previous studies have demonstrated that air-based design guidelines can be extrapolated to supercritical fluids [3,4]. Since the overall performance of a compressor is fundamentally determined by the aerodynamic efficiency of its individual blade sections, understanding the flow behavior around these blades is crucial. Building upon established similarity-based frameworks, this study focuses on a two-dimensional (2D) CFD analysis of the NACA 65-010 airfoil to specifically investigate how the incident angle of the blade affects aerodynamic efficiency [5,6]. By utilizing STAR-CCM+ to analyze the lift and drag coefficients, this research aims to provide fundamental data for optimizing S-CO_2 turbomachinery design under varying operational conditions.

2. Methodology

To analyze the aerodynamic performance of a compressor blade in the S-CO_2 cycle, the lift and drag coefficients were evaluated. For this purpose, a simplified 2D CFD analysis was conducted using a NACA airfoil in STAR-CCM+, as the cross-section of a compressor blade typically resembles an airfoil. The geometry of the NACA 65-010 airfoil was generated based on the data provided in the NACA report [7]. The airfoil shape is shown in Figure 1. To ensure the numerical accuracy and convergence of the simulation, a mesh independence test was systematically performed. The number of mesh elements was increased in increments of approximately 5,000 cells. Starting from a mesh count of 92,110 cells, the lift coefficient and drag coefficient values converged to 0.260 and 0.086, respectively. Since further increases in mesh density resulted in negligible changes to these aerodynamic coefficients, the mesh with 92,110 cells was selected as the final grid for all subsequent CFD analyses. The grid independence test was performed using the same representative operating condition for all mesh levels to ensure a consistent comparison.

To achieve high fidelity in the CFD results, the simulations were performed under steady-state conditions using a coupled implicit solver in STAR-CCM+. A steady-state approach was adopted because the present study focuses on the overall aerodynamic characteristics of the blade at fixed incident angles rather than on transient flow behavior.

The coupled implicit solver was selected to enhance numerical stability and convergence for compressible flow calculations involving large density differences between air and S-CO_2. The solution was considered converged when the residuals for all governing equations reached a criterion of 10^{-6}. In addition to the residual criterion, the lift and drag coefficients were monitored to ensure that the aerodynamic forces had also reached stable values.

The SST (Shear Stress Transport) k-omega turbulence model was selected to accurately predict the flow separation and wake characteristics of S-CO_2 at high Reynolds numbers. The SST k-omega model was adopted because it combines the near-wall accuracy of the k-omega formulation with the improved free-stream behavior of the k-epsilon formulation. To properly resolve the turbulent boundary layer, the mesh was refined near the airfoil surface to ensure a y+ value of less than 1.

The mesh was divided into a refined near-airfoil region and another region in order to accurately resolve the boundary layer and wake structure while maintaining computational efficiency.

Additional local refinement was applied near the leading edge, trailing edge, and wake region, where strong velocity and pressure gradients were expected.

The computational domain was discretized using a polyhedral mesh (often referred to as a "polygonal" approach in complex flow analysis) to ensure superior convergence and reduced numerical diffusion compared to standard tetrahedral meshes. A prism layer mesh was applied along the airfoil walls to capture the steep velocity gradients within the viscous sublayer.

Air and S-CO_2 properties were obtained using NIST REFPROP 10.0. The inlet and boundary conditions can be confirmed in Table 1. Initially, full similarity was considered to maintain identical aerodynamic performance between air and S-CO_2. However, when the Reynolds number is enforced to be preserved, it required an impractical reduction of the chord length to 0.0012 m due to the high density of S-CO_2. To ensure a realistic design, a partial similarity approach was adopted by releasing the Reynolds number constraint, allowing the chord length to be increased to 0.0885 m [4]. The airfoil surface was treated as a no-slip wall boundary, and the far-field boundaries were specified to minimize artificial blockage or confinement effects on the external flow.

These boundary conditions were selected to provide a consistent basis for comparing the aerodynamic behavior of air and S-CO_2 under the prescribed similarity framework.

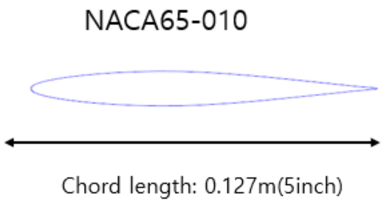

Fig.1. NACA 65-010 airfoil

Table1. Initial and boundary conditions

	Air	S-CO_2
Velocity [m/s]	28.956	0.158
Chord length [m]	0.127	0.127
Reynolds Number	245,000	245,000
Mach Number	0.083	0.0075
Pressure [Pa]	101325	$\mathbf{8 \times 10^{6}}$
Temperature [K]	300	315.15
Density [kg/m3]	1.1767	260.13
Viscosity [Pa-s]	1.79×10^{-5}	21.32×10^{-6}

3. Model Validation

To ensure the reliability of the numerical model, a validation process was conducted by comparing the simulation results with existing reference data [6]. The pressure coefficient is used for this validation because this can be visualized as velocity distribution in Figure 4. The pressure coefficient is defined by the following equation (1).

$$\boldsymbol{Pressure\ Coeff} = \frac{P_{total,upstream}-P_{static,local}}{P_{total,upstream}-P_{static,upstream}} \quad (1)$$

As shown in Figure 2, the CFD results correlate well with the reference data. This agreement demonstrates that the developed numerical model accurately captures the aerodynamic behavior of the S-CO_2 fluid within the specified operating range.

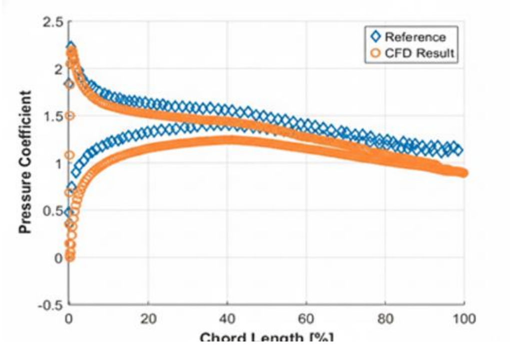

Fig. 2. Pressure Coefficient Comparison**4. Results**

3. Result

According to the results presented in Figure 3, the lift-to-drag ratio, which represents the overall aerodynamic efficiency, reaches its maximum value at an incident angle of approximately 8 degrees. This specific angle indicates the most efficient operating point for the compressor blade, where the highest lift is generated relative to the drag force.

Figure 3 shows that the lift-to-drag ratio reaches its maximum value near 8°, indicating the most aerodynamically efficient operating point within the investigated range. The S-CO_2 case exhibits a slightly higher C_L/C_D than the air case at this angle, while the overall aerodynamic trends of the two fluids remain broadly similar under the selected conditions. From the flow-field perspective, Figure 4 confirms that increasing the incident angle enlarges the separation region on the suction side and increases the wake velocity deficit, which is consistent with the increase in drag and the decrease in C_L/C_D. Since the present study is limited to a two-dimensional steady analysis and does not include a direct contour-based comparison for the air case, the physical significance of the observed difference near 8° should be examined further through future three-dimensional simulations with direct comparisons of the velocity and pressure fields.

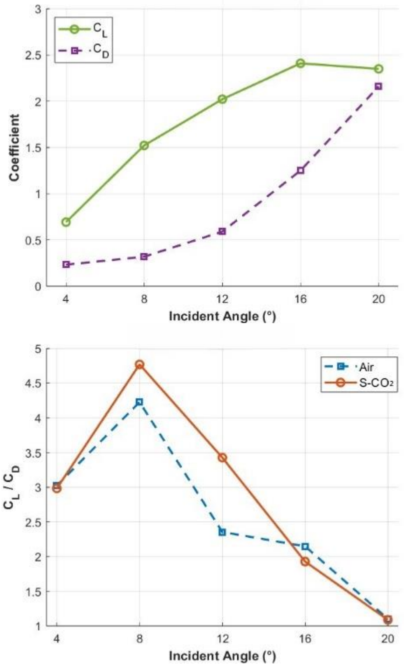

Fig. 3. Drag and Lift Coefficient & Lift-to-Drag Ratio

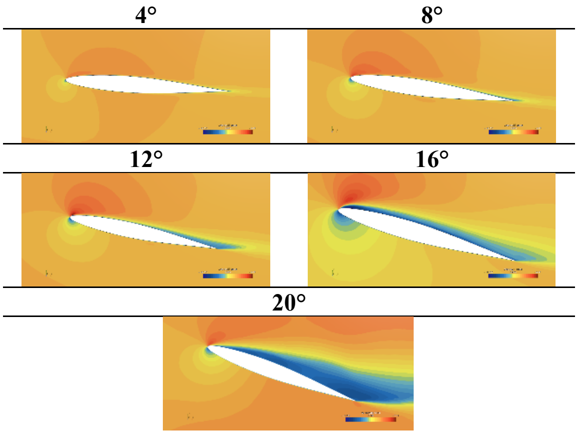

Fig. 4. Velocity Contour at Different Incident Angle

5. Summary and Conclusions

In summary, this study performed a two-dimensional CFD analysis to identify the optimal incident angle for an S-CO_2 compressor blade. Within the investigated range, the maximum lift-to-drag ratio occurred near 8 degrees, while the blade maintained relatively high lift up to approximately 16 degrees before stronger separation developed.

Compared with air, S-CO_2 showed broadly similar aerodynamic trends and a slightly higher C_L/C_D near 8 degrees; however, this limited difference is not sufficient to support a definitive claim of superior aerodynamic performance.

Instead, the present results indicate that S-CO_2 can achieve aerodynamic behavior comparable to that of air under the selected similarity and operating conditions. These findings provide fundamental data for estimating the stable operating range of S-CO_2 turbomachinery. Future work should include a more direct comparison of air and S-CO_2 velocity and pressure fields, together with three-dimensional effects such as tip leakage and secondary flows.

6. Future work

Since the present study was limited to a two-dimensional steady analysis, future work should extend the investigation to three-dimensional simulations to capture more realistic flow structures. A direct comparison of the velocity and pressure fields between air and S-CO_2 under three-dimensional conditions would help clarify whether the slightly higher C_L/C_D of S-CO_2 near 8° reflects a meaningful aerodynamic advantage or only a limited difference within the present analysis framework.

ACKNOWLEDGEMENT

This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIT) (No. RS-2025-25454059).

REFERENCES

[1] Y. Ahn, S. J. Bae, M. Kim, S. K. Cho, S. Baik, J. I. Lee, and J. E. Cha, "Review of Supercritical CO_2 Power Cycle Technology and Current Status of Research and Development," *Nuclear Engineering and Technology*, Vol. 47, No. 6, pp. 647-661, 2015.

[2] A. Romei, P. Gaetani, and G. Persico, "Computational fluid-dynamic investigation of a centrifugal compressor with inlet guide vanes for supercritical carbon dioxide power systems," *Energy*, Vol. 255, p. 124469, 2022.

[3] G. Kim, J. M. Baek, S. Lee, and J. I. Lee, "Preliminary Study of S-CO_2 Compressor Inlet Guide Vane (IGV) for Molten Salt Reactor," *Transactions of the Korean Nuclear Society Spring Meeting*, Jeju, Korea, May 22-23, 2025.

[4] C. Tello, A. Muñoz, D. Sánchez, T. Kipouros, and M. Savill, "Impact of Fluid Substitution on the Performance of an Axial Compressor Blade Cascade Working with Supercritical Carbon Dioxide," *J. Eng. Gas Turbines Power*, vol. 142, no. 1, p. 011019, Jan. 2020, doi: 10.1115/1.4045473.

[5] S. K. Cho, S. G. Kim, S. Baik, and J. I. Lee, "A Preliminary Study for Prediction of Supercritical CO_2 Compressor Surge Line," *Transactions of the Korean Nuclear Society Spring Meeting*, Jeju, Korea, May 18-19, 2017.

[6] Y. Jeong, S. K. Cho, and J. I. Lee, "An Aerodynamic Study of NACA Airfoil 6-Digits under CO_2 Environment by Two-dimensional Computational Fluid Dynamics," *Transactions of the Korean Nuclear Society Autumn Meeting*, Yeosu, Korea, October 25-26, 2018.

[7] Abbott, I. H., Von Doenhoff, A. E., and Stivers, L. S. Jr., "Summary of Airfoil Data," NACA Report No. 824, 1945.