

Parametric Optimization of the Aerodynamic Behavior of a Gas Turbine Blade : Influence of the Angle of Attack and the Reynolds Number

Amine Brahimi^{1*}, Mohammed Bey², Redha Rebhi³, Hakima Hachelef⁴

¹Scientific and Technology Research Direction- Cherchell Academy, Tipaza 42000 ,Algeria

* **Corresponding Author Email:** ami_brahimi@yahoo.fr - **ORCID:** 0000-0002-5247-6650

²Scientific and Technology Research Direction- Cherchell Academy, Tipaza 42000 ,Algeria

Email: bey_mohammed13@yahoo.fr - **ORCID:** 0000-0003-0833-3603

³LERM - Laboratory of Renewable Energies and Materials, University of Médéa, Médéa 26000, Algeria

Email: rebhi.redha@gmail.com - **ORCID:** 0000-0003-3019-9156

⁴Scientific and Technology Research Direction- Cherchell Academy, Tipaza 42000 ,Algeria

Email: hachelefhakima@yahoo.fr - **ORCID:** 0000-0002-5247-3540

Article Info:

DOI: 10.22399/ijcesn.5321

Received : 26 May 2026

Revised: 10 September, 2026

Accepted : 12 September 2026

Keywords

Turbomachinery aerodynamics

Numerical simulation (CFD)

Angle of attack

Reynolds number

Lift and drag coefficients

Abstract:

This study performs a Computational Fluid Dynamics (CFD) analysis of turbulent flow around a gas turbine blade, investigating the effects of angle of attack α and Reynolds number Re on aerodynamic performance. The objective was to quantify their impact on the lift C_L , drag C_D , and pressure C_p coefficients. Simulations used ANSYS CFX with the SST $k - \omega$ turbulence model and a refined mesh. Conditions spanned $\alpha = -10^\circ$ to $+40^\circ$ and $Re = 5000$ to 500000 , covering transitional to turbulent regimes. Both an isolated blade and a two-blade configuration were analyzed. Results show that α critically affects performance. C_L increases until a stall angle near 10° , then drops sharply due to flow separation, while C_D rises continuously. The Re is significant below 20000 (transitional flow), where C_L and C_D vary considerably. Above this threshold (fully turbulent), their sensitivity diminishes. The two-blade setup revealed aerodynamic interactions, with the downstream blade altering the upstream blade's effective flow, modifying local pressure and performance. The blade design requires optimizing α to balance lift and drag, and must account for Re -dependent effects, especially in transitional flow. Inter-blade interactions are also non-negligible for cascade performance.

1. Introduction

Gas turbines are now an essential technology in electricity generation, aircraft propulsion, and industrial applications, with performance that has steadily improved since their intensive development after World War II [1]. Their overall efficiency depends directly on the effectiveness with which the thermal energy of the combustion gases is converted into mechanical work at the turbine stages, where the blades play a crucial role [2]. These critical components are subjected to extreme operating conditions-severe thermal gradients, cyclic mechanical stresses, and corrosive environments-making them the most stressed elements of turbomachinery [3].

Optimizing the aerodynamic performance of the blades is therefore a major technological challenge

for improving the efficiency and reliability of gas turbines. Traditionally, this optimization relied primarily on experimental wind tunnel testing, but these approaches have significant limitations in terms of cost, flexibility, and the ability to accurately capture complex physical phenomena [4]. The emergence and development of numerical simulation methods, particularly computational fluid dynamics (CFD), have revolutionized this approach by enabling detailed and parametric analysis of flows in complex geometries [5].

The aerodynamic profiles of the blades, often derived from the well-characterized NACA series [6], are designed to generate an optimal compromise between lift and drag. The performance of these profiles is governed by several key parameters, including the angle of attack—defined as the angle between the chord of the profile and the upstream

flow direction—which directly influences the pressure distribution around the blade and therefore the resulting aerodynamic forces [7]. Previous studies have shown that the angle of attack significantly affects the lift coefficient up to a critical stall angle, beyond which performance degrades sharply [8].

In parallel, the Reynolds number—a dimensionless parameter characterizing the ratio between inertial and viscous forces—plays a fundamental role in flow structure. Reynolds' pioneering work [9] established the distinction between laminar and turbulent regimes, with important implications for heat and momentum transfer. In the context of turbomachinery, the Reynolds number influences boundary layer transition, separation phenomena, and vortex dynamics [10].

Reliable numerical modeling of these complex phenomena requires a rigorous approach to turbulence. Several modeling strategies exist, ranging from direct numerical simulation (DNS) [11], which is too expensive for most industrial applications, to large eddy simulation (LES) [12] and the more economical Reynolds-Averaged Navier-Stokes (RANS) approaches [13]. Among RANS models, the standard $k-\varepsilon$ model [14] and its RNG variant [15] have been widely used, but have limitations for flows with flow separation. The $k-\omega$ model [16], and particularly its SST (Shear-Stress Transport) version [17], which combines the advantages of the $k-\varepsilon$ and $k-\omega$ models, has proven particularly well-suited for predicting flows with adverse pressure gradients, common in turbomachinery [18].

In the specific field of gas turbines, previous numerical studies have focused on various aspects: the performance analysis of axial compressors [19], combustion optimization [20], and the prediction of blade life under thermomechanical loads [3]. However, a systematic parametric analysis combining the influence of the angle of incidence and Reynolds number on the aerodynamic characteristics of turbine blades, considering both isolated and interacting configurations, remains insufficiently documented in the literature.

Study Objectives

This study aims to fill this gap through a comprehensive numerical investigation of the aerodynamic behavior of a gas turbine blade. The specific objectives are :

1. To analyze the systematic effect of the angle of attack (ranging from -10° to $+40^\circ$) on the fundamental aerodynamic coefficients—lift C_L , drag C_D , and pressure C_p —as well as on the flow structure around the airfoil.
2. To evaluate the influence of the Reynolds number (coupling the range 5×10^3 to 5×10^5) on these

same parameters, identifying regime transitions and their implications for aerodynamic performance.

3. To compare the performance of a single blade with that of a two-blade configuration to characterize the upstream-downstream aerodynamic interactions and their effects on load distribution.

4. Validate a robust CFD simulation methodology based on the ANSYS CFX code and the SST turbulence model, producing reference data for the design and optimization of turbine blades.

This comprehensive parametric approach will establish quantitative correlations between design parameters and aerodynamic performance, thus contributing to the improvement of modern turbomachinery design tools.

2. Theoretical framework and methodology

2.1. Air foil Geometry and Study Conditions

This numerical study focuses on the aerodynamic analysis of a blade profile representative of industrial gas turbines. The basic geometry is inspired by NACA-type airfoils, widely documented for their performance in the internal flow of turbomachinery [6, 19]. The studied airfoil has a reference chord $c = 0.1$ m and a span sufficient to ensure a two-dimensional flow condition in the midsection, while allowing the modeling of three-dimensional effects at the tips.

Two distinct configurations were analyzed to isolate and then understand the interactions between blades:

1. Configuration A: A single blade placed at the center of a flow domain (figure 1).
2. Configuration B: Two identical blades arranged in tandem (one above the other), reproducing a simplified geometry of a turbine stage.

The computational domain, is a parallelepiped volume with dimensions of $4.356\text{ m} \times 3.3\text{ m} \times 0.3\text{ m}$. These dimensions were determined to ensure that the inlet and outlet boundaries are sufficiently far from the profile (at $20c$ and $30c$ respectively) to minimize the influence of boundary conditions on the flow near the blade, in accordance with established recommendations for CFD simulations of submerged objects [5]. The study conditions cover a representative operating range of regimes encountered in turbomachinery. The angle of attack α varies from -10° to $+40^\circ$ with principal pitch values of $-10^\circ, 0^\circ, 10^\circ, 20^\circ$, and 40° , allowing the study of normal operating regimes up to stall conditions. The Reynolds number Re , based on the airfoil chord and upstream velocity, varies from 5×10^3 to 5×10^5 , thus covering the transition regimes up to the fully established turbulent regime. The fluid considered is air, modeled as an incompressible ideal gas at ambient temperatures, with a density $\rho =$

1.225 kg/m^3 and a dynamic viscosity $\mu = 1.7894 \times 10^{-5} \text{ Pa}\cdot\text{s}$.

Table 1. Main geometric parameter

Parameter	Symbol	Value	Unit
Profile chord	c	0.100	m
Length	L_x	4.356	m
Height	L_y	3.300	m
Depth	L_z	0.300	m
Leading edge/entry distance	-	2.000	m (20c)
Trailing edge/exit distance	-	3.000	m (30c)

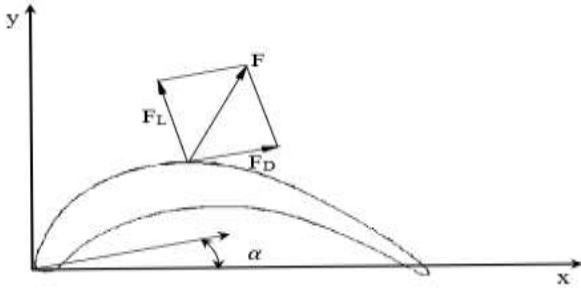


Figure 1. Geometry of the blade

2.2. Governing Equations

The flow of a Newtonian fluid is governed by the fundamental conservation equations. For turbulent, instantaneous, compressible flow, these equations are the Navier-Stokes equations. In this study, the flow is considered steady and incompressible (negligible density variations), an assumption valid for Mach numbers less than 0.3 [10]. The system of equations is then written as:

1. Continuity Equation (Conservation of Mass):

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (1)$$

Where u_i represents the components of the velocity vector (u, v, w) .

2. Equations for the conservation of momentum (Navier-Stokes):

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \rho g_i \quad (2)$$

with p the static pressure, μ the molecular dynamic viscosity and g_i the components of gravity (ρg_i neglected in this study).

3. Energy Conservation Equation:

For isothermal flow (the assumption used), the energy equation cannot be solved independently. The turbulent nature of the flow, confirmed by the high Reynolds numbers studied, makes **Direct Numerical Simulation** (DNS) resolution of the instantaneous equations prohibitively expensive.

The approach adopted is that of Reynolds-Averaged Navier-Stokes Equations (RANS), which consists of decomposing each instantaneous variable ϕ into a mean component $\bar{\phi}$ and a fluctuating component ϕ' ($\phi = \bar{\phi} + \phi'$). Applying this decomposition and a time average to equations (1) and (2) leads to the RANS system:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (3)$$

$$\frac{\partial(\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \overline{\rho u'_i u'_j} \right] \quad (4)$$

The term $\overline{\rho u'_i u'_j}$ is the Reynolds stress tensor, introducing six new unknowns (due to its symmetry). Therefore, closing the system requires a turbulence model that expresses these stresses as a function of the average flow variables.

2.3. Turbulence Modeling:

Choice and Justification of the $k-\omega$ SST Model

Turbulence modeling is critical for the accurate prediction of flows around lifting profiles, particularly in areas of adverse pressure gradient, boundary layer separation, and reattachment [13]. Several classes of models have been evaluated for their applicability to the case study:

- Single-equation models (e.g., Spalart-Allmaras) are economical but often limited to specific flows for which they have been calibrated [14].
- Two-equation models of the $k-\varepsilon$ type (standard, RNG, realizable) is robust and widely used for fully turbulent flows far from the walls. However, they have difficulty accurately predicting separations under strong adverse pressure gradients and require wall functions to handle the region near the walls [14, 15].
- The standard Wilcox $k-\omega$ model performs remarkably well in the viscous sublayer region and for low Reynolds number flows, but it exhibits excessive sensitivity to free turbulence conditions at the domain inlet [16].

To overcome these limitations, the Menter $k-\omega$ Shear-Stress Transport (SST) model [17] was selected for this study. This hybrid model combines the advantages of the $k-\omega$ (near the wall) and $k-\varepsilon$ (far the wall) models via an F_1 mixing function. Its main strengths for turbine blade simulation are:

1. Accuracy in flows with adverse pressure gradients: The SST model incorporates a Reynolds stress-principle limiter, preventing overestimation of turbulent viscosity in areas of high shear, thus improving boundary layer separation prediction [18].
2. Robustness and relative independence from inlet conditions: Unlike the standard $k-\omega$ model, the

SST model is less sensitive to the specified value of ω (specific dissipation rate) at the inlet.

3. Extensive validation in turbomachinery: The model has demonstrated its ability to accurately predict complex flows in compressors and turbines, including jet-wake interactions [18].

The SST model is defined by two transport equations, one for the turbulent kinetic energy k and the other for the specific dissipation rate ω :

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho U_j k)}{\partial x_j} = \tilde{P}_k - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right] \quad (5)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho U_j \omega)}{\partial x_j} = \alpha \rho S^2 - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \frac{\rho \sigma_{\omega_2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (6)$$

With $\tilde{P}_k$ the limited production of turbulent kinetic energy, $\mu_t = \rho k / \omega$ the turbulent viscosity, S the invariant of the strain rate tensor, and F_1 the mixing function which is 1 near the walls (activating the $k - \omega$ model) and 0 far from the walls (activating a $k - \varepsilon$ type formulation). The constants $\alpha, \beta, \beta^*, \sigma_k, \sigma_\omega$ are also interpolated by this function. This formalism allows a robust transition between the treatment of the internal and external zones of the boundary layer, essential for correctly capturing the flow physics around the complete profile.

2.4. Numerical Method and Solution Procedure

The numerical solution of the closed RANS system of equations using the SST $k - \omega$ turbulence model was performed using the commercial software ANSYS CFX 19.0. This solver uses a coupled finite volume approach, considered robust for internal and external flows in turbomachinery [5, 19]. The finite volume method ensures the discrete conservation of physical quantities (mass, momentum) on each mesh element, an essential property for simulation accuracy. The pressure-velocity coupling is handled by the Coupled Algebraic Multigrid algorithm implemented in ANSYS CFX. Unlike segregated algorithms (e.g., SIMPLE, PISO), this algorithm solves the momentum and continuity equations as a single, coupled system, which significantly accelerates convergence for steady flows, particularly those exhibiting strong interaction between velocity and pressure fields.

The convective terms of the transport equations are discretized using the High-Resolution Scheme. This scheme is essentially second-order accuracy, but it incorporates a slope limiter that ensures numerical stability by preventing oscillations at strong discontinuities, while preserving accuracy in low-gradient regions. The diffusive terms are discretized with a second-order centered scheme.

The convergence of the solution is monitored by tracking the normalized residuals (RMS) for each solved equation (continuity, momentum, k, ω). The simulation is considered converged when these residuals reach a value less than 10^{-5} and the overall aerodynamic coefficients (lift and drag) stabilize with variations of less than 0.1% over 100 consecutive iterations. A maximum of 1000 iterations was set for each simulation case, a threshold sufficient to achieve convergence in all cases studied.

2.5. Mesh Generation and Boundary Conditions

Mesh quality is a critical factor for the accuracy of CFD simulations, particularly in areas with high physical variable gradients [5]. A hybrid unstructured mesh was generated using the ANSYS Meshing module. The region near the blade tip, where significant velocity and pressure gradients are expected, was refined with prismatic elements to accurately capture the boundary layer. The remainder of the domain was meshed with tetrahedra for greater geometric flexibility.

The resolution of the boundary layer is critical for the SST turbulence model. The parameter y^+ (the dimensionless height of the first cell) was kept close to 1 across the entire blade surface. This value is recommended to ensure that the first node lies within the viscous sublayer, allowing the SST model to directly solve the physics up to the wall without relying on wall functions. To achieve this condition, a progressive geometric growth (ratio ~ 1.2) was applied from the wall towards the core of the flow. Figure 2 (to be provided) illustrates the mesh refinement in the vicinity of the blade. A mesh independence study was conducted on the single-blade configuration with $\alpha = 10^\circ$ and $Re = 15,000$.

Table 2. Mesh Characteristics.

Parameter	Configuration A (1 blade)	Configuration B (2 blades)
Total number of nodes	22 667	38 000
Total number of elements	116 972	195 000
Prismatic elements	15 000	25 000
Tetrahedral elements	101 972	170 000
Number of prismatic layers	15	15
Average y^+ on the blade	0.8 - 1.2	0.8 - 1.2
Growth ratio	1.2	1.2

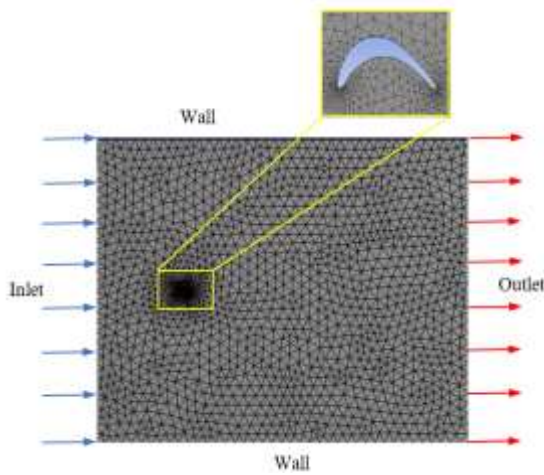


Figure 2. Mesh and boundary conditions

Three mesh resolution levels were tested: Coarse Mesh ($\approx 60k$ elements), Medium Mesh ($\approx 117k$ elements, used), and Fine Mesh ($\approx 230k$ elements). The variation in the lift coefficient C_L between the medium and fine meshes was less than 1.5%, and that of the drag coefficient C_D was less than 2%. The medium mesh was therefore selected as offering an optimal compromise between accuracy and computation time.

3. Results and discussion

3.1. Influence of the Angle of Attack (Configuration A - 1 blade)

Figure 3 illustrates the significant impact of the angle of attack on the flow structure around the blade profile. For a negative angle of attack ($\alpha = -10^\circ$), the maximum velocity is located on the upper surface (extrados), generating a pronounced asymmetric velocity gradient. At zero angle of attack ($\alpha = 0^\circ$), the velocity distribution becomes almost symmetrical, with comparable values on both sides of the profile, indicating a minimum lift condition. Conversely, for a positive angle of attack ($\alpha = +10^\circ$), a complete reversal of the phenomenon is observed: the velocity accelerates sharply on the lower surface (intrados), while a large area of reduced velocity, characteristic of incipient separation, appears on the upper surface (extrados) from the leading edge to the trailing edge.

The static pressure distribution directly follows these trends. At $\alpha = -10^\circ$, a strong overpressure develops on the lower surface (intrados) near the leading edge, while a moderate underpressure develops on the upper surface (extrados). At $\alpha = 0^\circ$, the pressure is maximum at the stagnation point located precisely at the leading edge, with a generally symmetrical distribution. At $\alpha = +10^\circ$, the pressure configuration is radically reversed: a

strong underpressure develops on the lower surface and a marked overpressure develops on the upper surface, thus creating the pressure gradient necessary to generate high positive lift.

Turbulent kinetic energy, for its part, reveals the losses associated with each configuration. For negative and zero angles, turbulence levels remain low in the core of the flow, with only a slight increase in the near wake. In contrast, at $\alpha = +10^\circ$, intense turbulent production is observed in the separated upper surface area and in the widened wake, indicating substantial energy losses. This drastic increase in turbulence at high positive angles of attack confirms the proximity of the stall angle and highlights the fundamental trade-off between lift generation and aerodynamic efficiency.

3.2. Evolution of the Pressure Coefficient

Figure 4 shows the evolution of the pressure coefficient C_p along the chord of the airfoil for five distinct angles of attack ($-10^\circ, 0^\circ, 10^\circ, 20^\circ, 40^\circ$), highlighting the crucial influence of this parameter on the aerodynamic load distribution. For all angles studied, a positive pressure peak is consistently observed at the leading edge, corresponding to the stagnation point where the flow splits. This maximum value of C_p decreases progressively as the angle of attack increases in the positive direction, indicating a change in the local flow dynamics at the point of impact.

Comparative analysis of the curves reveals a fundamentally asymmetrical behavior depending on the airfoil's orientation. For a negative angle of attack ($\alpha = -10^\circ$), the pressure drop is more pronounced on the upper surface than on the lower surface, thus generating negative lift. At zero angle of attack ($\alpha = 0^\circ$), the pressure distribution is relatively symmetrical, although a slight difference persists due to the potential camber of the airfoil, with overall higher C_p values. Conversely, for positive angles of attack ($\alpha = 10^\circ, 20^\circ, 40^\circ$), a complete reversal of the phenomenon occurs: the lower surface now exhibits the most pronounced low pressure, while the C_p values on the upper surface become less negative, even positive at the highest angles. More specifically, the angle $\alpha = 10^\circ$ exhibits the most favorable pressure distribution for lift generation, with a maximum ΔC_p gradient between the two surfaces of the airfoil. Beyond this value, particularly for $\alpha = 20^\circ$ and 40° , a progressive collapse of this gradient is observed, accompanied by a rise in the pressure curves along the entire chord. This behavior is characteristic of aerodynamic stall: the early separation of the boundary layer on the upper surface reduces the effective low-pressure area, thus decreasing the lift

generated despite the increased angle. These results clearly identify the optimal angle of attack for this

airfoil, located around 10° , beyond which aerodynamic performance degrades significantly.

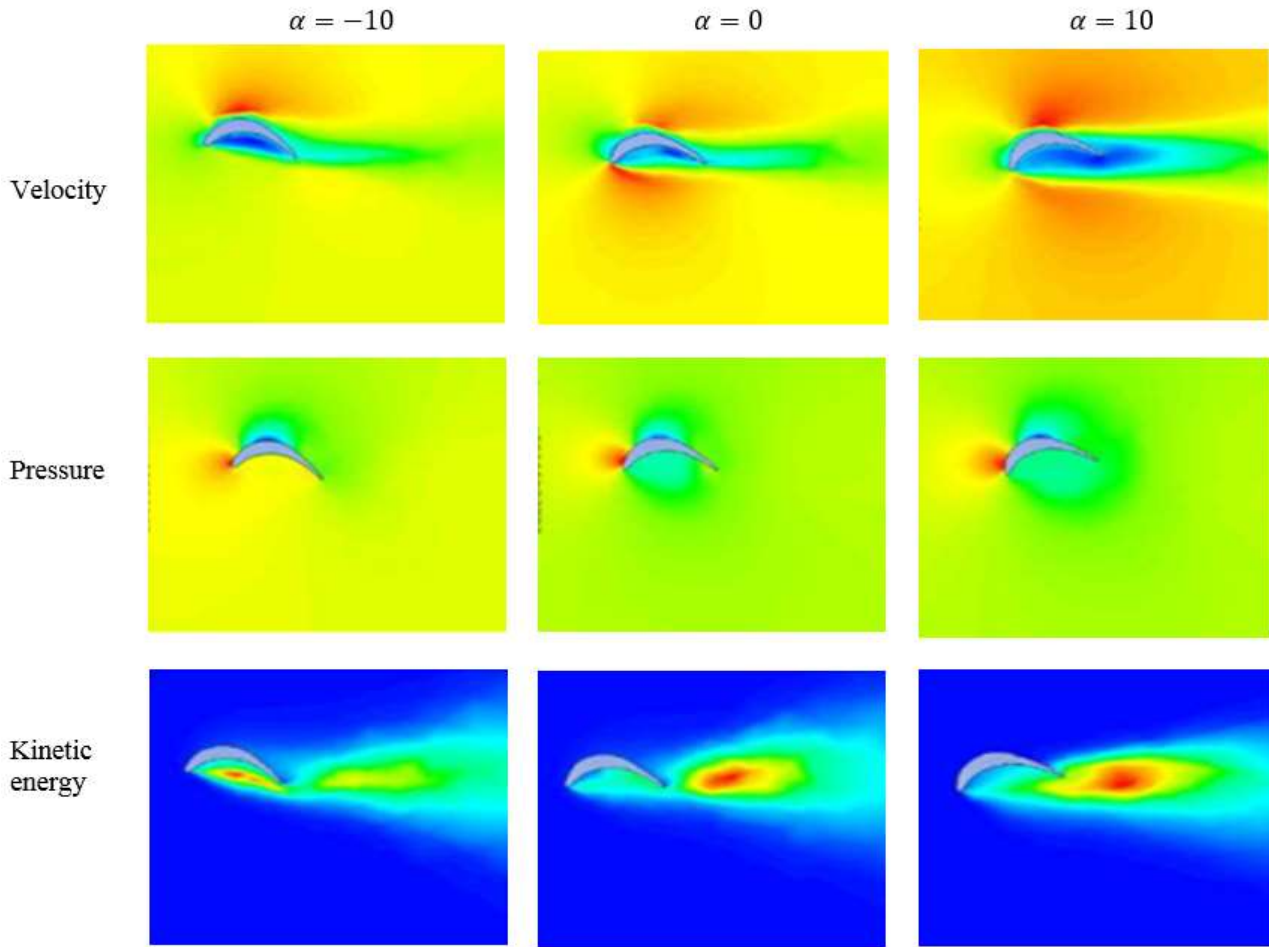


Figure 3. Contours of velocity, pressure and kinetic energy for different angles α

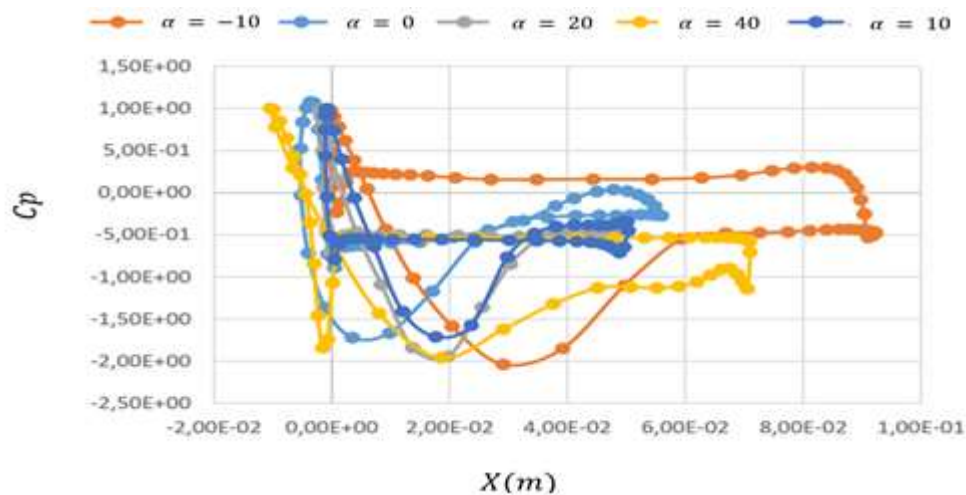


Figure 4. Pressure coefficient around the profile at different angles of incidence

3.3. Evolution of Drag and Lift Coefficients as a Function of the Angle of Attack

Figure 5 illustrates the simultaneous evolution of the lift coefficient C_L and drag coefficient C_D over a range of angles of attack from -10° to 40° ,

revealing the fundamental relationships that govern the aerodynamic performance of the airfoil. The lift coefficient exhibits a nearly linear increase between -10° and 10° , reaching a pronounced maximum around 10° before dropping sharply at higher angles of attack. This marked peak at 10° corresponds to

the angle of maximum lift, beyond which the stall phenomenon occurs, characterized by a massive separation of the boundary layer on the upper surface. The abrupt drop in C_L between 10° and 20° , followed by its stabilization at a reduced value at 40° , confirms the critical nature of this aerodynamic threshold.

In parallel, the drag coefficient C_D shows a contrasting but complementary evolution. While maintaining relatively low and stable values for negative and zero angles, C_D begins a gradual increase at $\alpha = 10^\circ$, which accelerates considerably at higher angles. This exponential growth of drag beyond the maximum lift angle is explained by the expansion of the separation zone and the corresponding widening of the wake, which generate increased pressure losses and frictional forces. The divergence observed between the C_L and C_D curves from 10° onward highlights the critical aerodynamic trade-off: the marginal lift gains obtained by increasing the angle of attack beyond the optimum are quickly negated by the surge in drag losses.

The combined analysis of these two parameters allows us to identify the optimal operating window for the airfoil, located between 0° and 10° , where the C_L/C_D ratio (aerodynamic efficiency) is maximized. The particularly high C_L value at $\alpha = 10^\circ$, combined with a still moderate increase in C_D , suggests that this angle represents the most efficient operating point for applications where lift generation is paramount. These results underscore the crucial importance of precise angle-of-attack tuning in the design and operation of turbine blades, where each degree of deviation from the optimum can translate into significant energy efficiency losses.

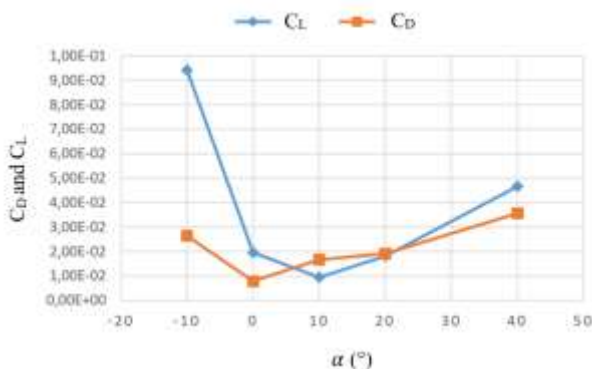


Figure 5. Drag coefficient and lift as a function of angle α ($-10^\circ, 0^\circ, 10^\circ, 20^\circ, 40^\circ$)

3.4. Evolution of the pressure coefficient C_p around the airfoil at different Reynolds numbers

Figure 6 shows the influence of the Reynolds number on the pressure coefficient distribution around the blade profile at zero angle of attack $\alpha = 0^\circ$, revealing the significant impact of flow regimes

on aerodynamic loads. For the lowest Reynolds numbers $Re = 5000$ and 10000 , corresponding to laminar-turbulent transition regimes, C_p distributions exhibit less pronounced gradients, particularly at the leading edge where the pressure peak is attenuated. These curves reflect a flow where dominant viscous effects limit fluid acceleration on the airfoil surfaces, thus reducing characteristic pressure drops.

As the Reynolds number increases $Re = 15000, 20000, 40000$, a gradual shift towards more contrasting pressure distributions is observed. The leading edge pressure peak intensifies, reaching higher maximum C_p values, while the pressure drops on the upper and lower surfaces become more pronounced. This transformation is explained by the relative decrease in viscous influence compared to inertial forces, allowing for greater local accelerations and stronger pressure gradients. The boundary layer, thinner at high Reynolds numbers, adheres better to the airfoil surface, thus generating pressure distributions closer to potential theory.

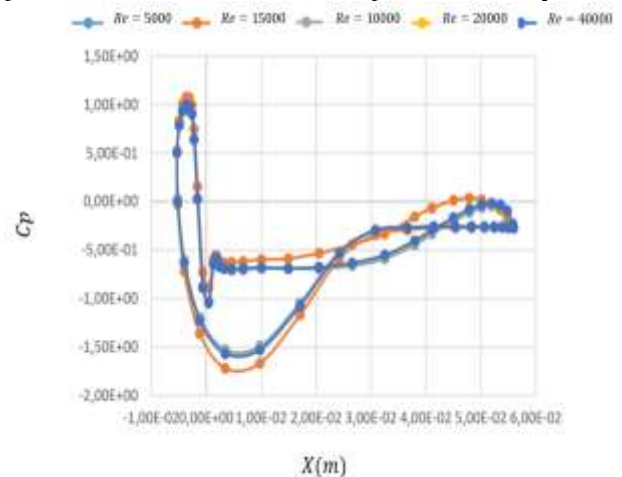


Figure 6. Pressure coefficient C_p around the profile at different Reynolds numbers

Figure 6 particularly highlights a convergence phenomenon: for $Re = 20000$ and 40000 , the pressure curves almost perfectly overlap across the entire chord, indicating that the flow has reached a fully turbulent regime where the pressure distribution becomes independent of subsequent increases in the Reynolds number. This stabilization suggests the existence of a critical threshold, located around $Re = 20000$ for this airfoil, beyond which the aerodynamic characteristics become insensitive to variations in the similarity parameter. These results have direct implications for turbine blade design, demonstrating that performance optimization must consider transition regimes ($Re < 20,000$), where sensitivity to the Reynolds number is high, and fully turbulent regimes ($Re > 20,000$), where this influence becomes negligible, differently.

3.5. Evolution of Drag and Lift Coefficients as a Function of Reynolds Number

Figure 7 shows the influence of the Reynolds number on the drag coefficient C_D and the lift coefficient C_L at zero angle of attack $\alpha = 0^\circ$, revealing distinct and complementary behaviors between these two fundamental aerodynamic parameters. The lift coefficient exhibits a rapid and almost linear increase between $Re = 5000$ and $Re = 15000$, rising from a minimum value to a pronounced maximum, before beginning a gradual decrease at higher Reynolds numbers $Re = 20000$ and 40000). This bell-shaped curve reflects the transition between different boundary layer regimes: at low Reynolds numbers (5000 – 15000), the reduction in boundary layer thickness and the improvement in flow adhesion to the airfoil surface allow for greater fluid acceleration over the upper surface, thus increasing the pressure gradient and the generated lift. The decrease observed beyond $Re = 15000$ suggests that the flow, becoming fully turbulent, generates increased friction losses that partially offset the gains associated with the thinning of the boundary layer.

Conversely, the drag coefficient shows an overall decreasing trend over the studied range, with a significant decrease between $Re = 5000$ and $Re = 15000$, followed by relative stabilization at higher values. This initial reduction in drag is explained by the gradual transition from a flow with a strong viscous component to a regime where inertial forces predominate, thus limiting the size of the separation zone and the associated pressure drag. The stabilization of C_D beyond $Re = 15000$, coupled with the slight increase observed at $Re = 40000$, indicates the establishment of a balance between the reduction in form drag and the concomitant increase in friction drag due to increased turbulence.

Comparing the two curves highlights an aerodynamic optimum located around $Re = 15000$, where the airfoil simultaneously exhibits maximum lift and minimum drag, thus optimizing its aerodynamic efficiency (C_L/C_D ratio). These results demonstrate that turbine blade performance cannot be considered monotonic with respect to the Reynolds number, but rather exhibits a specific optimum linked to the complex interactions between the laminar-turbulent transition, boundary layer adhesion, and friction losses. This analysis underscores the importance of controlling flow conditions in turbomachinery design to exploit the most favorable operating regimes. Figure 8 illustrates the impact of the angle of attack on the flow structure in the two-bladed configuration,

revealing complex aerodynamic interactions absent in the single-bladed case.

For a negative angle of attack $\alpha = -10^\circ$, the maximum velocity is concentrated on the upper surface of the upper blade (blade 01), while the lower blade (blade 02) exhibits a more disturbed flow with an extended low-velocity zone on its lower surface. This vertical asymmetry generates a strongly differentiated pressure distribution: a marked low pressure appears on the upper surface of blade 01, while a high pressure dominates around blade 02, thus creating a lift imbalance between the two elements.

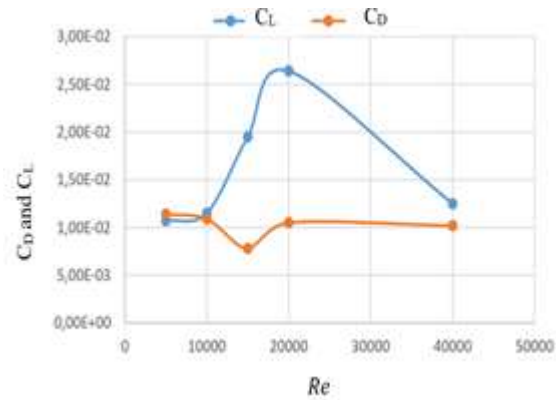


Figure 7. Drag coefficient and lift as a function of Reynolds number

3.6. Influence of the Angle of Attack (Configuration B - 2 Blades)

At zero angle of attack $\alpha = 0^\circ$, the contours reveal some symmetrization of the flow, although subtle differences remain. The velocity over the upper surface of blade 01 decreases slightly compared to the previous case, while the flow around blade 02 becomes more regular. The pressure is distributed more evenly between the two blades, although a residual gradient indicates that aerodynamic interference persists even in the absence of geometric angle of attack. The turbulent kinetic energy, generally low, shows a slight increase in the combined wake of the two blades, signaling the onset of wake interactions.

For a positive angle of attack $\alpha = +10^\circ$, a complete reversal of the phenomenon observed at $\alpha = -10^\circ$ is observed, but with an intensity amplified by the aerodynamic masking effect. blade 02, located downstream, is now subjected to the flow disturbed by blade 01: a large area of reduced velocity and high turbulence develops over its upper surface, characteristic of separation caused by interference. blade 01 exhibits a pronounced acceleration on its lower surface and an intense low-pressure area, generating maximum lift but at the cost of degraded flow over the downstream blade. The pressure

between the two blades now shows a gradient reversed compared to the case $\alpha = -10^\circ$, with the strongest low-pressure areas on the lower surfaces. Energy analysis reveals that the interaction between the two blades significantly amplifies turbulence production, particularly at non-zero angles of attack. At $\alpha = +10^\circ$, an extensive turbulent cloud is observed, extending from the wake of blade 01 to partially cover blade 02, generating aerodynamic losses far greater than those of a single blade. These visualizations clearly demonstrate that in a multi-blade configuration, the angle of incidence not only controls the individual performance of each element, but also modulates the nature and intensity of their interactions, with direct consequences on the overall efficiency and flow stability in a turbine stage.

3.7. Evolution of the Pressure Coefficient for the Two-Blade Configuration

The comparative analysis of the pressure coefficients for the two blades in the superimposed configuration reveals significant aerodynamic interactions that substantially modify the pressure distributions compared to the case of a single blade. For blade 01 (upstream), the influence of the angle of attack on the pressure coefficient C_p follows trends similar to those of the isolated case, but with attenuated amplitudes.

At $\alpha = -10^\circ$, the pressure gradient between the lower and upper surfaces, although present, is less pronounced than in the isolated case, indicating that the presence of blade 02 downstream disrupts the establishment of a potential flow around the upstream blade. The pressure peak at the leading edge of blade 01 (Figure 9-a), consistently present for all angles, shows a reduction in intensity, particularly for negative and zero angles.

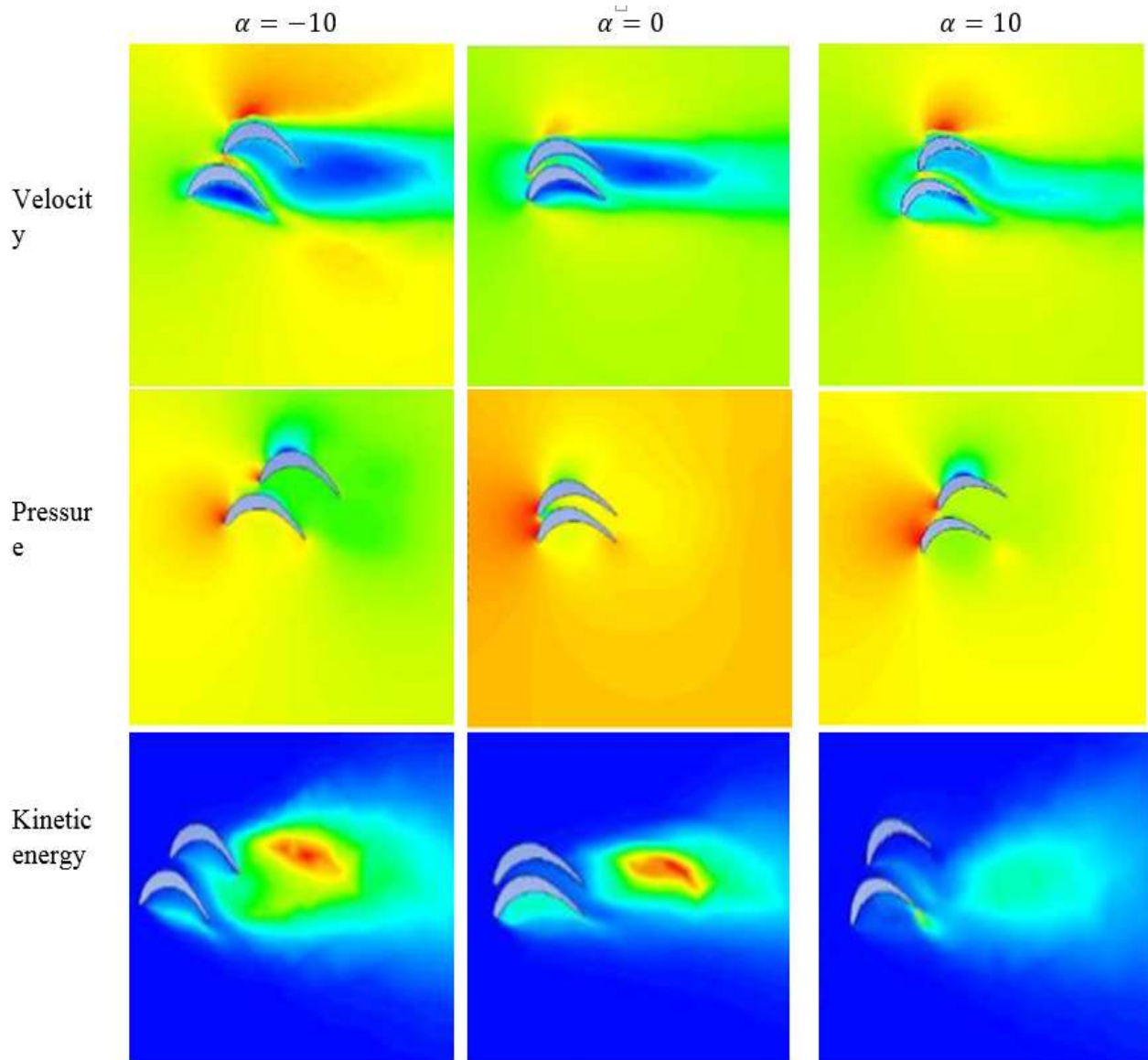


Figure 8. Contours of velocity, pressure and kinetic energy for different angles α

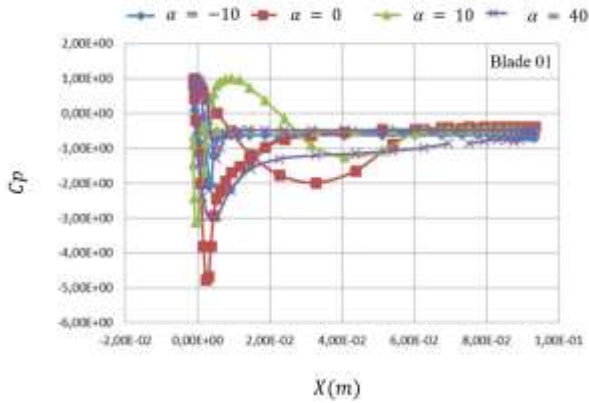


Figure 9-a. Pressure coefficient around the profile at different angles of incidence ($-10^\circ, 0^\circ, 10^\circ, 40^\circ$)

For the downstream edge (blade 02), the C_p distributions exhibit radically different characteristics, reflecting the aerodynamic shading effect. At $\alpha = -10^\circ$ and 0° , the pressure curves of blade 02 almost overlap, with overall higher (less negative) C_p values than those of blade 01 on the corresponding surfaces. This behavior is explained by the deceleration of the flow before reaching blade 02, a direct consequence of the disturbance generated by blade 01. The kinetic energy available to create low pressure areas on blade 02 is therefore reduced, limiting its capacity to generate lift (Figure 9-b). The case $\alpha = +10^\circ$ exhibits a particular behavior where the pressure coefficient of blade 02 shows a notable increase along the chord compared to blade 01. This inversion of the pressure gradient suggests that blade 02 is operating under degraded aerodynamic conditions, probably due to the separated flow from blade 01 impacting its leading edge. At $\alpha = 40^\circ$, the most striking phenomenon, a substantial increase in C_p values is observed on blade 02 compared to blade 01, confirming that the downstream blade is fully affected by the turbulent wake of the upstream blade, with a flattened pressure distribution characteristic of separated flow. These results demonstrate that in a two-blade configuration, the angle of attack cannot be optimized independently for each element. Finding an optimal angle requires considering the system as a whole, balancing the individual performance of each blade with the unfavorable interactions they generate between them. The configuration at $\alpha = 0^\circ$ appears to be the best compromise, minimizing interference while maintaining relatively favorable pressure distributions on both blades.

3.8. Drag and Lift Coefficients of Blade 01 as a Function of the Angle of Attac

Figure 10 shows the evolution of the aerodynamic coefficients of blade 01 (upper blade) in the two-blade configuration. The lift coefficient C_{L1} reaches a pronounced maximum at $\alpha = 10^\circ$ before decreasing significantly at higher angles, following

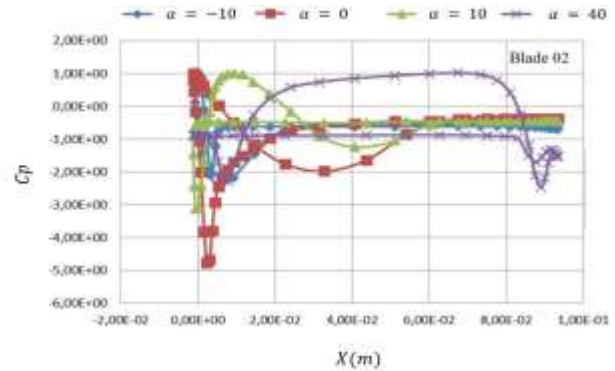


Figure 9-b. Pressure coefficient around the profile at different angles of incidence ($-10^\circ, 0^\circ, 10^\circ, 40^\circ$)

a similar but less pronounced trend compared to the isolated case. The drag coefficient C_{D1} exhibits a biphasic behavior characteristic of interacting systems: it reaches relative minima at extreme angles $\alpha = -10^\circ$ and 10° while a marked peak appears at $\alpha = 20^\circ$. This distribution suggests that blade 01, although positioned upstream, is influenced by the pressure field perturbed by the presence of blade 02, thus modifying its aerodynamic response compared to the isolated case. The angle $\alpha = 10^\circ$ appears to be the optimal operating point for this blade, combining high lift with moderate drag.

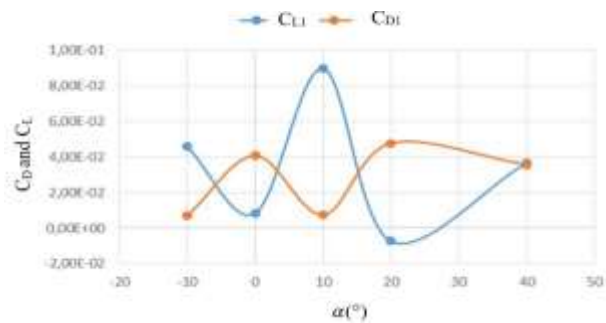


Figure 10. Drag coefficient and lift as a function of the angle of attack ($-10^\circ, 0^\circ, 10^\circ, 20^\circ, 40^\circ$)

3.9. Drag and Lift Coefficients of Blade 02 as a Function of the Angle of Attack

Figure 11 illustrates the distinct behavior of blade 02 (lower blade), revealing pronounced aerodynamic interference phenomena. Unlike blade 01, the lift coefficient C_{L2} has its minimum at $\alpha = 10^\circ$ and reaches a maximum at $\alpha = 20^\circ$, demonstrating a response lag characteristic of one blade operating in the wake of another. The drag coefficient C_{D2}

follows an opposite trend to that observed for blade 01, with a minimum at $\alpha = 0^\circ$ and increased values at extreme angles, particularly at $\alpha = -10^\circ$. These reversed trends confirm that the 02 blade operates in fundamentally different aerodynamic conditions from its upstream counterpart, fully experiencing the effects of the disturbed flow generated by the 01 blade. The downstream position exposes this blade to an unfavorable trade-off between lift and drag, limiting its individual performance.

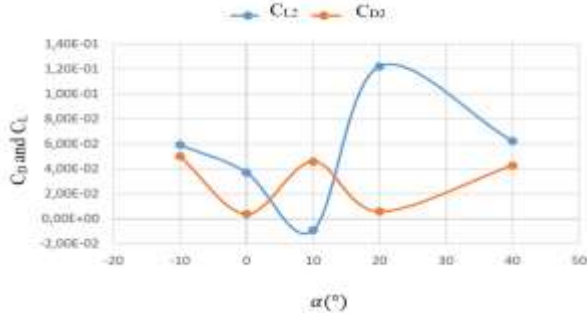


Figure 11. Drag coefficient and lift as a function of the angle of attack ($-10^\circ, 0^\circ, 10^\circ, 20^\circ, 40^\circ$)

3.10. Comparison of the drag coefficients of the two blades as a function of the angle of attack

Figure 12 summarizes the phenomenon of aerodynamic antagonism between the two blades by superimposing the evolution of their drag coefficients. Clearly, almost mirror-image trends are observed: when C_{D1} is minimal $\alpha = -10^\circ$ and 10° , C_{D2} exhibits relatively high values, and conversely, when C_{D1} reaches its maximum at $\alpha = 20^\circ$, C_{D2} is at its lowest level. This opposing relationship reflects a transfer of aerodynamic losses within the coupled system: optimizing the flow around one blade often comes at the expense of the other. At $\alpha = 20^\circ$, the strong stall of blade 01 generates an extensive turbulent wake which, paradoxically, reduces the adverse pressure gradients on blade 02, thus decreasing its drag while degrading overall efficiency. This analysis reveals that in a multi-blade configuration, drag minimization cannot be addressed independently for each element, but must be considered a systemic optimization problem requiring trade-offs between interacting components.

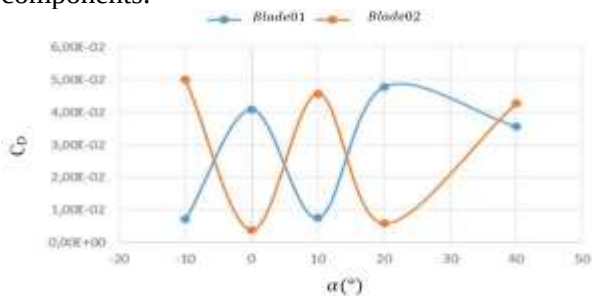


Figure 12. Drag coefficient as a function of the angle of attack ($-10^\circ, 0^\circ, 10^\circ, 20^\circ, 40^\circ$)

Figure 13 illustrates the opposing behavior of the lift coefficients of the two blades in a superimposed configuration, highlighting an aerodynamic compensation phenomenon within the coupled system. The curve of blade 01 C_{L1} shows a pronounced peak at $\alpha = 10^\circ$ followed by a sharp drop, characteristic of a classic stall, while that of blade 02 C_{L2} shows the opposite trend with a minimum at $\alpha = 10^\circ$ and a delayed maximum at $\alpha = 20^\circ$. This mirror effect reveals that the two blades do not operate under independent flow conditions, but interact to redistribute the generated lift according to the imposed angle of attack.

Comparative analysis shows that at $\alpha = 10^\circ$, where blade 01 reaches its maximum performance $C_{L1} \approx 0.08$, blade 02 operates under degraded conditions with its minimum lift $C_{L2} \approx 0.03$. This situation corresponds to a configuration where blade 01, individually optimized, generates an organized wake that adversely disturbs the flow incident on blade 02, thus reducing its lift-producing capacity. Conversely, at $\alpha = 20^\circ$, when Blade 01 stalls and its lift collapses $C_{L1} \approx 0.04$, blade 02 paradoxically benefits from more favorable conditions, reaching its maximum lift $C_{L2} \approx 0.055$. This reversal is explained by the modification of the wake structure: the largely separated flow coming from the blade 01 creates a low-pressure zone which, although dissipative, can induce favorable local pressure gradients on certain parts of the blade 02.

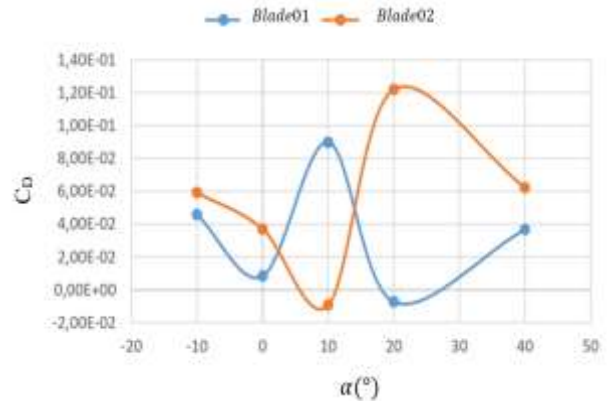


Figure 13. Lift coefficient as a function of the angle of incidence ($-10^\circ, 0^\circ, 10^\circ, 20^\circ, 40^\circ$)

3.11. Effect of Reynolds Number

Figure 14 illustrates the significant impact of the Reynolds number on the flow structure in the two-bladed configuration, revealing distinct transitions in the aerodynamic interaction mechanisms. For the lowest Reynolds numbers $Re = 5000$ and 10000), corresponding to the laminar-turbulent transition regimes, the velocity contours show pronounced

separation zones on both blades, particularly on the upper surface of blade 01 and the lower surface of blade 02. These early separations generate broad, diffuse wakes that interact strongly, creating an extensive region of reduced velocity between the two profiles. The pressure exhibits attenuated gradients, with less pronounced depressions on the acceleration surfaces, while the turbulent kinetic energy remains localized mainly in the immediate wakes, indicating turbulence production limited primarily to the separation zones. As the Reynolds number increases $Re = 15000, 20000, 40000$, a progressive transformation towards more organized flow structures is observed. The separation zones are

considerably reduced, particularly on blade 01, which now exhibits a more tightly bound flow on its upper surface. The velocity contours become more contrasted, with steeper gradients near the blades and more compact wakes. This evolution is accompanied by an intensification of the low-pressure areas on the convex surfaces of both profiles, generating larger pressure gradients conducive to lift production. Turbulent kinetic energy, meanwhile, shows a significant redistribution: while it decreases in the separation zones (due to improved boundary layer adhesion), it increases in regions of high shear, particularly in the inter-blade space and in the combined wakes.

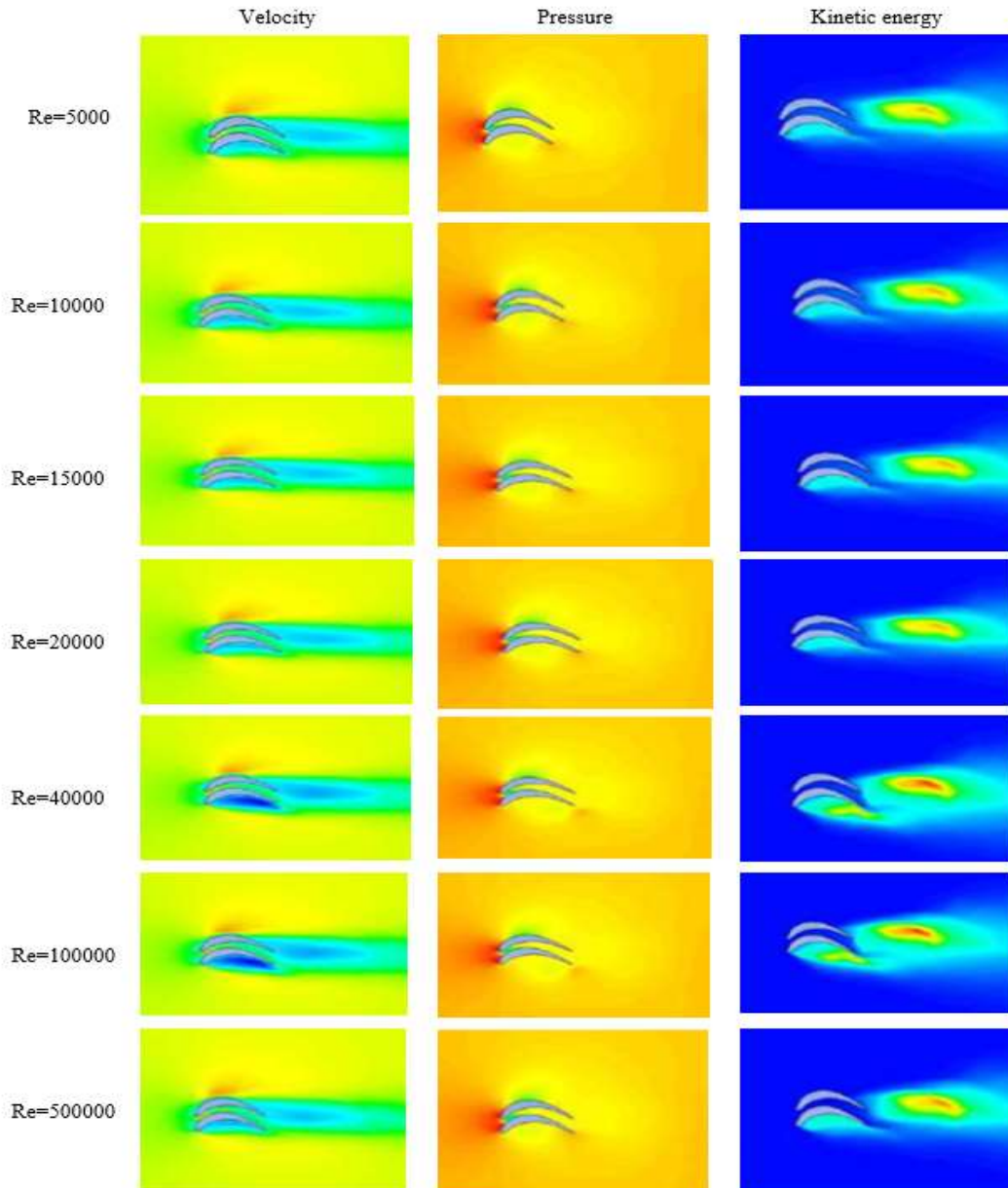


Figure. 14 Contours of velocity, pressure and kinetic energy for different Reynolds numbers

At the highest Reynolds numbers $Re = 100000$ and 500000 , the flow reaches a fully turbulent regime characterized by a quasi-stabilized structure. The

velocity contours show a high degree of similarity between these two cases, with well-defined areas of maximum acceleration and narrow, organized

wakes. The pressure distribution becomes more symmetrical between the two blades, suggesting a reduction in the aerodynamic shadowing effects observed at low Reynolds numbers. Turbulent kinetic energy is now concentrated primarily in thin mixing layers at the interfaces between the wakes and the free flow, as well as in the high shear regions downstream of the trailing edges.

This visual sequence demonstrates that the Reynolds number controls not only the individual characteristics of each blade but also the very nature of their interactions. At low Reynolds numbers, the interactions are dominated by the interference of the large, separate wakes, generating significant losses. At intermediate Reynolds numbers, a significant improvement is observed due to reduced separation. At high Reynolds numbers, the system reaches an optimal regime where interactions become predictable and less dissipative, allowing for better exploitation of pressure gradients for lift generation.

3.12. Evolution of the pressure coefficient around the airfoil for different Reynolds numbers (Two-blade configuration)

Figure 15 shows the influence of the Reynolds number on the pressure coefficient distribution C_p for the two-blade configuration, revealing distinct behaviors between the upstream and downstream blades, as well as a critical transition in the aerodynamic response of the coupled system. For blade 01, a progressive evolution of the pressure curves is observed with increasing Reynolds number: at $Re = 5000$, the C_p distribution exhibits attenuated gradients, with a reduced pressure peak at the leading edge and moderate pressure drops along the surfaces, characteristic of a flow with strong viscous influence where the thick boundary layer limits local accelerations. As the Reynolds number increases to 10000 and then 20000 , the pressure peak at the leading edge intensifies significantly, while the pressure drops on the surfaces become more pronounced, particularly on the lower surface, indicating improved flow adhesion and a reduction in early separation effects. For blade 02, positioned downstream, the C_p distributions exhibit fundamentally different characteristics and increased sensitivity to Reynolds number. At $Re = 5000$, the pressure curve is remarkably flattened over the entire chord, with C_p values close to zero, which reflects a flow strongly degraded by interactions with the wake of blade 01. This situation corresponds to a decoupled flow regime where the downstream blade operates in a zone of greatly reduced velocity, limiting its ability to generate significant pressure gradients. As the Reynolds number increases $Re = 10,000$ then $20,000$, a

progressive restoration of the pressure characteristics of blade 02 is observed, with the appearance of a pressure peak at the leading edge and the development of better-defined depressions, although still of smaller amplitude than those of blade 01. This improvement is explained by the reduction in the thickness and intensity of the wake of blade 01 at higher Reynolds numbers, allowing blade 02 to partially recover more favorable flow conditions. Direct comparison between the two graphs highlights a phenomenon of differential convergence: while the C_p curves for blade 01 tend to converge with increasing Reynolds number (particularly between $Re = 10000$ and 20000), those of blade 02 show more pronounced differences across the entire studied range. This observation suggests that blade 02 remains more sensitive to Reynolds number variations than its upstream counterpart, as it experiences not only the direct effects of this parameter on its own flow, but also its indirect effects through changes in the characteristics of the incident wake. The fact that the pressure distributions for $Re = 10000$ and 20000 on blade 02 do not completely overlap indicates that the system has not yet reached a regime fully independent of the Reynolds number in the studied range, unlike what was observed for the isolated blade.

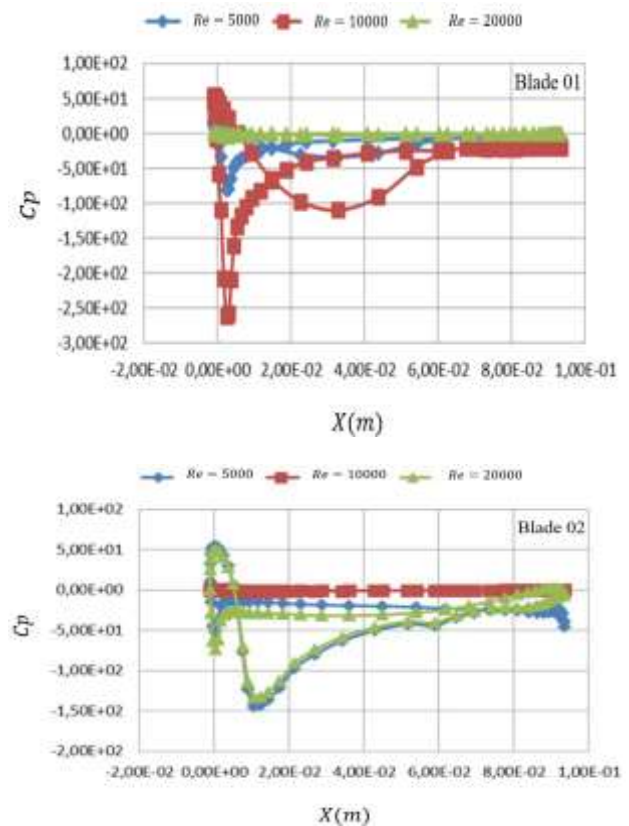


Figure. 15 Pressure coefficient C_p around the profile at different Reynolds numbers

3.13. Effect of Reynolds Number on Lift and Drag Coefficients

Figure 16 illustrates the evolution of the aerodynamic coefficients of blade 01 (upstream blade) under the increasing influence of the Reynolds number. The lift coefficient C_{L1} shows a moderate but continuous increase, rising from approximately 0.038 at $Re=5000$ to 0.042 at $Re=40000$. This gradual increase reflects the improved aerodynamic efficiency of the upstream blade as viscous effects decrease, allowing for better boundary layer adhesion and more favorable pressure gradients. Concurrently, the drag coefficient C_{D1} exhibits a steady decrease, evolving from 0.032 to 0.026 over the same Reynolds number range. This reduction in drag, more pronounced than the increase in lift, demonstrates that blade 01 benefits significantly from the transition to a more inertial flow regime, where friction and separation losses are reduced. The simultaneous improvement of these two parameters results in a marked increase in the lift-to-drag ratio C_{L1}/C_{D1} , confirming that the upstream blade's overall performance is optimized with increasing Reynolds number, independently of interactions with the downstream blade.

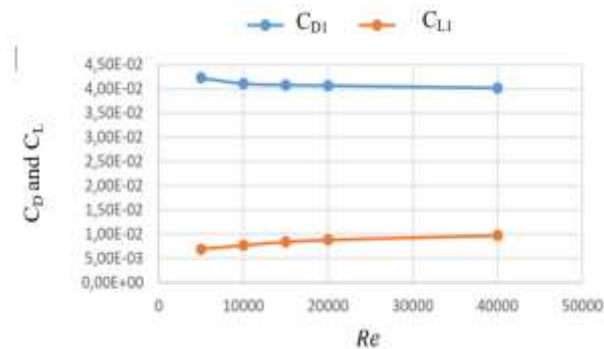


Figure. 16 Drag coefficient and lift as a function of Reynolds number

Figure 17 reveals the increased sensitivity of blade 02 (downstream blade) to Reynolds number variations, with more marked changes than those observed for the upstream blade. The lift coefficient C_{L2} increases moderately from 0.035 to 0.038, following a similar but slightly less pronounced trend than that of C_{L1} . In contrast, the drag coefficient C_{D2} undergoes a dramatic reduction, dropping from 0.035 to just 0.015 between $Re=5000$ and $Re=40000$. This decrease of more than 50% contrasts sharply with the 20% drop observed for C_{D1} , indicating that the downstream blade benefits disproportionately from the improved flow conditions at high Reynolds numbers. This differential evolution is explained by the dual influence of the Reynolds number on blade 02: it benefits both from the improvement in its own

boundary layer characteristics and, above all, from the qualitative transformation of the wake of blade 01, which becomes more organized and less disruptive at high Reynolds numbers.

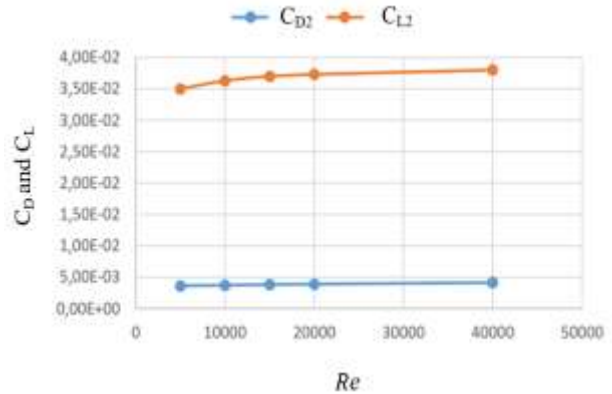


Figure. 17 Drag coefficient and lift as a function of Reynolds number.

Figure 18 highlights a critical transition in the aerodynamic interactions between the two blades, marked by a crossover point around $Re=15000$. For Reynolds numbers below this threshold, $C_{D2} > C_{D1}$ is consistently observed, confirming that the downstream blade bears the brunt of the losses generated by unfavorable interactions in the laminar-turbulent transition regime. This situation reverses dramatically above $Re=15000$, where C_{D2} becomes smaller than C_{D1} , indicating that the downstream blade now benefits from a drag advantage.

This reversal reflects a change in the nature of the interactions: at low Reynolds numbers, the wide and diffuse wake of blade 01 creates a zone of strong disturbance that penalizes blade 02; at high Reynolds numbers, this same wake becomes structured into a more organized jet that can even induce a beneficial "guiding" effect for the downstream blade. This crossover point thus identifies the threshold at which the system transitions from a regime of destructive interference to one of potentially synergistic interactions.

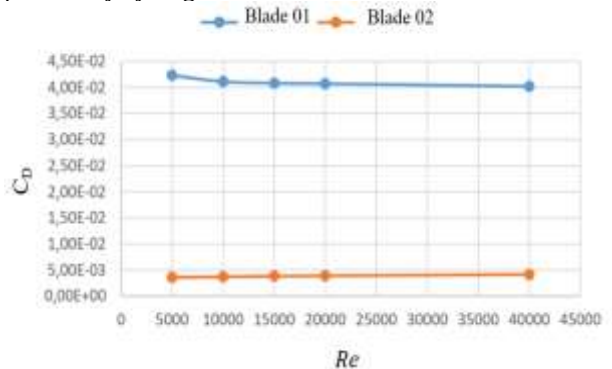


Figure. 18 Drag coefficient as a function of Reynolds number

Figure 19 shows a more regular evolution of the lift coefficients, without the crossover observed for the drag coefficients. Over the entire Reynolds number range studied, C_{L1} remains consistently higher than

C_{L2} , with a relatively constant difference of approximately 0.004. This stable hierarchy indicates that the positional advantage of blade 01—which benefits from undisturbed incident flow—persists regardless of the flow regime. However, the two curves have similar slopes, suggesting that both blades respond similarly to increasing Reynolds number in terms of lift generation. This similarity in behavior contrasts with the divergent trends observed for drag, revealing that the Reynolds number influences lift mechanisms (primarily linked to pressure gradients) and drag mechanisms (strongly dependent on wake interactions and separation phenomena) differently. The stability of the gap between the two curves suggests that optimizing lift performance in a multi-blade stage would require differentiated geometric modifications rather than simply adjusting the overall flow conditions.

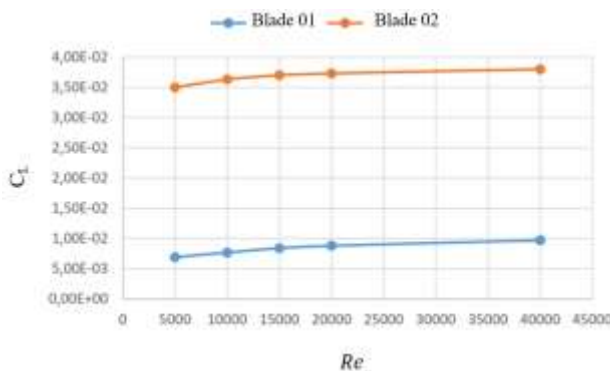


Figure 19 Lift coefficient as a function of Reynolds number

4. Conclusion

This in-depth numerical study of turbulent flow around a gas turbine blade, considered in isolation and then in a two-blade configuration, has allowed us to systematically quantify the determining influence of two fundamental parameters: the angle of attack and the Reynolds number. The results obtained by CFD simulation with the SST turbulence model in ANSYS CFX have highlighted complex aerodynamic mechanisms with direct implications for the design and optimization of turbomachinery. Analysis of the isolated blade confirmed classical aerodynamic laws while precisely quantifying them for the studied airfoil. The optimal angle of attack was identified at $\alpha = 10^\circ$, corresponding to the maximum lift coefficient $C_L \approx 0.032$ before aerodynamic stall. Beyond this threshold, the rapid deterioration of performance, characterized by a sharp drop in lift and a surge in drag, underscores the critical importance of precise control of blade orientation during operation. The influence of the Reynolds number revealed a marked behavioral

transition: for $Re < 20000$, the aerodynamic coefficients exhibit significant sensitivity, whereas above this threshold, pressure distributions and performance tend to stabilize, indicating the establishment of a fully turbulent regime with characteristics almost independent of the Reynolds number. The study of the two-blade configuration revealed interaction phenomena of greater complexity, with major implications for the design of complete turbine stages. The antagonistic behavior observed between the upstream blade (01) and the downstream blade (02)—where optimizing the performance of one often comes at the expense of the other—demonstrates that the design approach cannot be limited to the individual optimization of airfoils, but must incorporate a systemic view of aerodynamic interactions. The discovery of a critical crossover point around $Re = 15000$, where the dominance relationship between the drags of the two blades reverses, identifies a crucial operational threshold: below this point, the downstream blade suffers marked disadvantages; above it, the system evolves towards a regime of potentially synergistic interactions.

Visualizations of the velocity, pressure, and turbulent kinetic energy fields provided a thorough physical understanding of the underlying mechanisms, showing how the upstream blade wake structure evolves qualitatively with the Reynolds number, transitioning from a broad, diffuse wake (highly disruptive) to a more organized jet (potentially beneficial) for the downstream blade.

Perspectives and recommendations:

1. Optimization of blade pitch angles: The results support differential blade pitch within a stage, with angles slightly below the individual optimum for the upstream blades, to minimize the negative impact on the downstream blades.
2. Flow regime control: Operating the turbines in the $Re > 20000$ range allows for stable and repeatable performance, less sensitive to variations in operating conditions.
3. Adaptive Geometric Design: The difference in Reynolds number sensitivity between upstream and downstream blades suggests that a differentiated geometry (profiles, cambers) could optimize the overall performance of the stage.
4. Future Studies: Further investigations incorporating transient and thermal effects, as well as the presence of a third stator stage, could refine these conclusions, bringing them closer to the actual operating conditions of gas turbines.

Author Statements:

- **Ethical approval:** The conducted research is not related to either human or animal use.

- **Conflict of interest:** The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper
- **Acknowledgement:** The authors declare that they have nobody or no-company to acknowledge.
- **Author contributions:** The authors declare that they have equal right on this paper.
- **Funding information:** The authors declare that there is no funding to be acknowledged.
- **Data availability statement:** The data that support the findings of this study are available on request from the corresponding author. The data are not publicly available due to privacy or ethical restrictions.

References

- [1] Mattingly, J. D. (1996). Elements of gas turbine propulsion. McGraw-Hill.
- [2] Dixon, S. L., & Hall, C. (2013). Fluid mechanics and thermodynamics of turbomachinery. Butterworth-Heinemann.
- [3] Schobeiri, M. T. (2018). Gas turbine design, components and system design integration. Springer.
- [4] Kim, K., Samad, A., & Benini, E. (2019). Optimization of fluid machinery. John Wiley & Sons.
- [5] Anderson, J. D. (2010). Fundamentals of aerodynamics. McGraw-Hill.
- [6] Jacobs, E. N., & Pinkerton, R. M. (1936). Tests in the variable-density wind tunnel of related airfoils having the maximum camber unusually far forward (NACA Report No. 537). NACA.
- [7] Moran, J. (2003). Introduction to theoretical and computational aerodynamics. Dover Publications.
- [8] Bertin, J. J., & Cummings, R. M. (2014). Aerodynamics for engineers. Pearson.
- [9] Reynolds, O. (1883). An experimental investigation of the circumstances which determine whether the motion of water shall be direct or sinuous, and of the law of resistance in parallel channels. Philosophical Transactions of the Royal Society, 174, 935–982. <https://doi.org/10.1098/rstl.1883.0029>
- [10] White, F. M. (2011). Fluid mechanics. McGraw-Hill. Moin, P., & Mahesh, K. (1998). Direct numerical simulation: A tool in turbulence research. Annual Review of Fluid Mechanics, 30(1), 539–578. <https://doi.org/10.1146/annurev.fluid.30.1.539>
- [11] Sagaut, P. (2006). Large eddy simulation for incompressible flows. Springer.
- [12] Wilcox, D. C. (2006). Turbulence modeling for CFD (3rd ed.). DCW Industries.
- [13] Launder, B. E., & Spalding, D. B. (1974). The numerical computation of turbulent flows. Computer Methods in Applied Mechanics and Engineering, 3(2), 269–289. [https://doi.org/10.1016/0045-7825\(74\)90029-2](https://doi.org/10.1016/0045-7825(74)90029-2)
- [14] Yakhot, V., & Smith, L. M. (1992). The renormalization group, the ϵ -expansion and derivation of turbulence models. Journal of Scientific Computing, 7(1), 35–61. <https://doi.org/10.1007/BF01060210>
- [15] Wilcox, D. C. (1988). Reassessment of the scale-determining equation for advanced turbulence models. AIAA Journal, 26(11), 1299–1310. <https://doi.org/10.2514/3.10041>
- [16] Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 32(8), 1598–1605. <https://doi.org/10.2514/3.12149>
- [17] Bardina, J. E., Huang, P. G., & Coakley, T. J. (1997). Turbulence modeling validation, testing, and development (NASA Technical Memorandum No. 110446). NASA.
- [18] Cumpsty, N. A. (2004). Compressor aerodynamics. Krieger Publishing Company.
- [19] Lefebvre, A. H., & Ballal, D. R. (2010). Gas turbine combustion (3rd ed.). CRC Press. <https://doi.org/10.1201/9781420086058>