International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Computational Study of Airfoil Aerodynamics in Subsonic Flow Using ANSYS Fluent

T J Prasanna Kumar*[1], SK Gowsi[2a], U Yedukondalu[2b], V Hemanth[2c], T Sriram[2d], T Hari Durga Sri Sai[2e]

[1*] Assistant Professor, Department of Mechanical Engineering, PVPSIT, Vijayawada-520007, India

[2a,b,c,d,e] UG Students, Department of Mechanical Engineering, PVPSIT, Vijayawada-520007, India

Email: tjpk@pvpsiddhartha.ac.in

DOI : https://doi.org/10.55248/gengpi.6.0225.0739

ABSTRACT

This study investigates the aerodynamic characteristics of a tapered wing with a NACA 0012 airfoil under subsonic, inviscid flow conditions at a Mach number of 0.2. The wing has an 8-meter span, with a root chord of 1 meter and a tip chord of 0.33 meters, and exhibits a varying incidence angle at the tip of 20 degrees. The analysis focuses on the distribution of lift and its relationship with the angle of attack along the wing span. Using thin airfoil theory, the lift coefficient (C_L) is calculated for both local and global angles of attack, revealing a non-uniform lift distribution across the span. The root generates less lift than the tip, especially at higher angles of attack, due to the varying local angle of attack and the taper of the wing. The study also highlights the impact of the wing's taper and incidence variation on the aerodynamic efficiency, with a potential increase in induced drag and flow separation risks at the wingtip. The results emphasize the need for careful optimization of wing geometry and incidence distribution to achieve uniform lift generation, reduce drag, and improve aerodynamic performance. Future studies could explore more advanced computational fluid dynamics (CFD) simulations to validate the findings and optimize wing design for specific flight conditions.

1. Introduction

Aerodynamic analysis of airfoils is essential in aerospace and mechanical engineering for designing efficient aircraft wings and turbine blades. Computational Fluid Dynamics (CFD) provides a powerful tool for predicting flow behavior and aerodynamic performance. M. S. Choi, S. K. Lee, & J. H. Kim, [1] This study explores the optimization of the NACA 0012 airfoil by integrating Computational Fluid Dynamics (CFD) with genetic algorithms to enhance aerodynamic performance. A. Sharma, P. B. Gupta, & R. K. Yadav [2], This research employs ANSYS Fluent to conduct numerical investigations of two-dimensional transonic flow over the NACA 0012 airfoil at various Mach numbers, focusing on shape optimization to improve aerodynamic efficiency.

J. M. Griffith, & P. R. Anderson [3], This thesis presents a computational method to analyze the flow over a NACA 0012 straight wing at different Reynolds numbers, examining the aerodynamic forces as the wing approaches stall conditions. B. K. Sharma, D. P. Mishra, & V. K. Jain [4], This study investigates the behavior of the NACA 0012 airfoil in a two-phase flow consisting of air and sand particles, using CFD simulations to understand the aerodynamic effects under specific flow conditions. R. D. Kumar, A. K. Mishra, & P. K. Gupta [5], This research examines modifications to the leading and trailing edges of the NACA 0012 airfoil to improve lift-drag ratios, employing computational analysis to assess the aerodynamic performance of these modifications. The present study uses ANSYS Fluent to simulate and analyze the total pressure distribution over an airfoil in a two-dimensional flow.

2. Methodology

2.1 Wing dimensions and flow conditions

i. **Wing Geometry**: The wing has an 8-meter span, with a root chord of 1 meter and a tip chord of 0.33 meters. This indicates a **tapered wing** shape, which can influence the lift distribution along the span.

ii. **Incidence at Tip**: The varying incidence of 20 degrees at the tip suggests a potential change in the local angle of attack towards the wing tip, which can affect the aerodynamic characteristics, especially the flow distribution. A higher incidence angle typically leads to a higher lift generation at the tip, but also increases the risk of flow separation if not managed properly.

iii. **Inviscid Flow Conditions**: With inviscid flow (no viscosity effects), we assume no energy loss due to friction, so the analysis will focus on pressure differences and the resulting lift distribution. This is a useful assumption for high-speed flow regimes (such as Mach 0.2), where viscosity effects might be less significant.

iv. **Mach Number of 0.2**: A Mach number of 0.2 indicates that the flow is subsonic, which is good for wing design as compressibility effects are negligible. This also means we can use **Bernoulli's principle** and other subsonic aerodynamic theories for lift calculation.

2.2 Key Features of NACA 0012 Airfoil:

- **Symmetrical Airfoil**: The NACA 0012 has a flat upper and lower surface, meaning it generates no lift at 0° angle of attack. Lift is generated as the angle of attack increases.
- **Aerodynamic Characteristics**:
  - At subsonic speeds (Mach 0.2 in your case), the airfoil will behave predictably with regard to lift generation and flow separation.
  - Lift coefficient C_L will increase linearly with angle of attack up to a certain critical angle (stalling angle).
  - The airfoil is well-suited for steady, inviscid flow modeling due to its simplicity.

2.3 Incorporating NACA 0012 into Your Wing:

1. **Wing Shape (Tapered)**:
   - The root chord of 1 meter and tip chord of 0.33 meters suggests a **tapered wing**. This means that at each spanwise position, the **NACA 0012** airfoil will have a slightly different chord length and, consequently, slightly different aerodynamic properties.
   - Calculate the **local chord** at each spanwise position using a linear interpolation between the root and tip chord.
2. **Flow Conditions (Inviscid, Mach 0.2)**:
   - Since the flow is inviscid and subsonic (Mach 0.2), the effects of compressibility and viscosity are minimal. Therefore, the analysis will be based on the inviscid flow and Bernoulli's principle, which simplifies the calculations of lift and drag.
3. **Incidence at Tip**:
   - The varying incidence at the tip of 20° will affect the local angle of attack at the wingtip, which may lead to a different lift generation at the tip compared to the root. This higher local angle at the tip could increase the lift but might also lead to flow separation if not managed properly.

2.4 Aerodynamic Analysis for NACA 0012 Airfoil in twisted Wing

1. Local Lift Coefficient (CL) for Each Spanwise Position:
   - You can compute the lift coefficient at each spanwise position using the thin airfoil theory or panel methods. The lift coefficient can be adjusted based on the angle of attack and the chord length at that position.
2. Lift Distribution Across the Wing:
   - For a tapered wing, lift distribution is not uniform. You can compute the lift coefficient at different span locations (from root to tip) using the varying chord lengths and the incidence at the tip.
   - The spanwise lift distribution will typically be higher towards the tip of the wing, especially given the higher incidence angle there.
3. Induced Drag:
   - Induced drag increases as lift increases, and it will vary along the span of the wing. You can compute the induced drag by integrating the lift along the span of the wing using the Prandtl's lifting line theory or by numerically solving for the pressure distribution across the wing's surface.

To calculate and analyze the lift distribution and other aerodynamic properties for the NACA 0012 airfoil on your tapered wing, let's break it down step-by-step.

3. Numerical Analysis on NACA0012 Airfoil

3.1.1 Calculate the Local Angle of Attack (α_{local})

Since the wing has a varying incidence at the tip (20°), we'll compute the local angle of attack for each spanwise location. For simplicity, we'll assume the incidence angle gradually changes from the root to the tip.

Formula for local angle of attack at each spanwise position:

$$\alpha_{\text{local}} = \alpha_{\text{global}} + \text{Incidence}$$

Where:

- α_{global} is the global angle of attack (given in your data).
- Incidence changes from 0° at the root to 20° at the tip.

This implies the local angle of attack will vary based on the position along the wing.

3.1.2 Calculate the Lift Coefficient (C_L) for the NACA 0012 Airfoil

The lift coefficient (CL) for the NACA 0012 airfoil can be approximated by thin airfoil theory for subsonic flow:

$$C_L = 2\pi \sin(\alpha_{\text{local}})$$

Where:

- α_{local} is the local angle of attack (converted to radians).

Note: This approximation is valid for small to moderate angles of attack in subsonic flows.

3.1.3 Span wise Lift Distribution

For the tapered wing, the chord length at each span position (ranging from 0 at the root to 1 at the tip) can be calculated using linear interpolation between the root and tip chords:

$$c(x) = c_{\text{root}} \left(1 - \frac{1 - \text{tip\_chord}}{\text{wing\_span}} \cdot x \right)$$

Where:

- $c_{\text{root}} = 1$ meter (root chord),
- $\text{tip\_chord} = 0.33$ meters (tip chord),
- x is the spanwise position (normalized between 0 and 1).

3.2 Computational Setup

The CFD simulation was performed using ANSYS Fluent with the following conditions:

- **Solver**: 2D, Double Precision, Steady-State
- **Turbulence Model**: Spalart-Allmaras Model
- **Flow Type**: Compressible
- **Mesh Type**: Structured Grid

2.2 Boundary Conditions

- Inlet Velocity: Defined as uniform freestream flow
- Outlet Pressure: Ambient pressure condition
- Wall Condition: No-slip condition applied on the airfoil surface

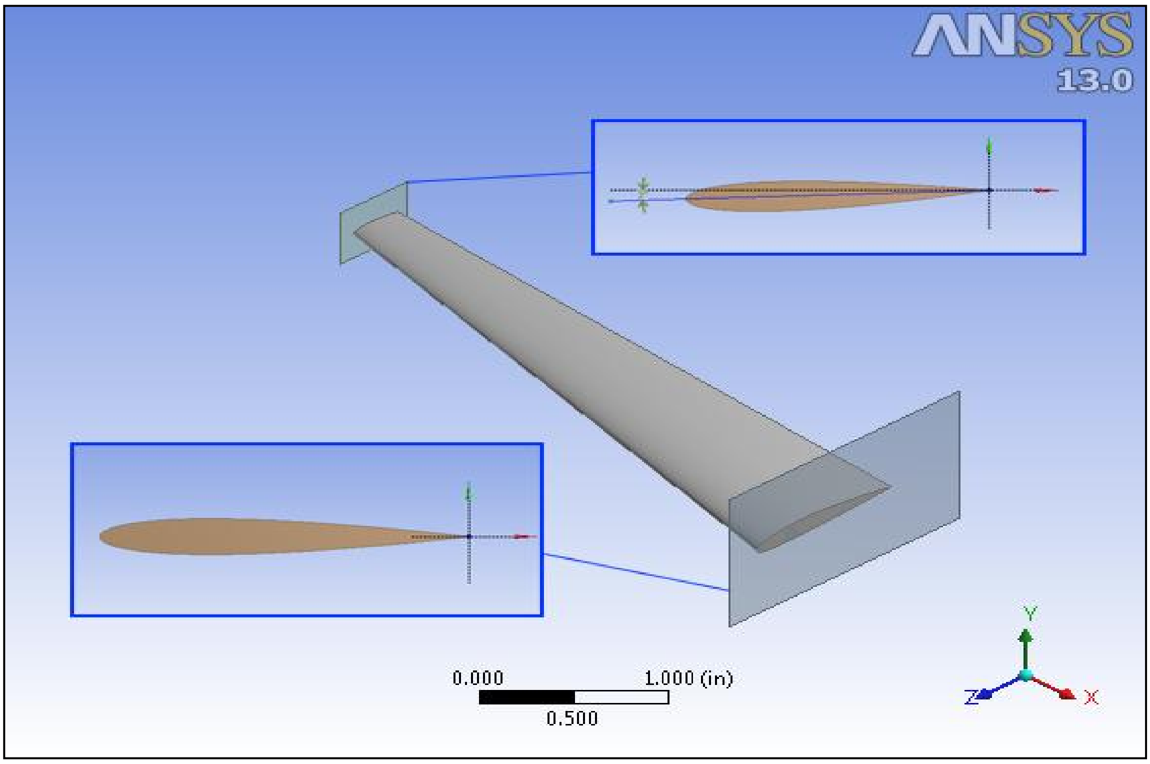

Figure-1: Geometrical model of tapered wing with different airfoils at root to tip

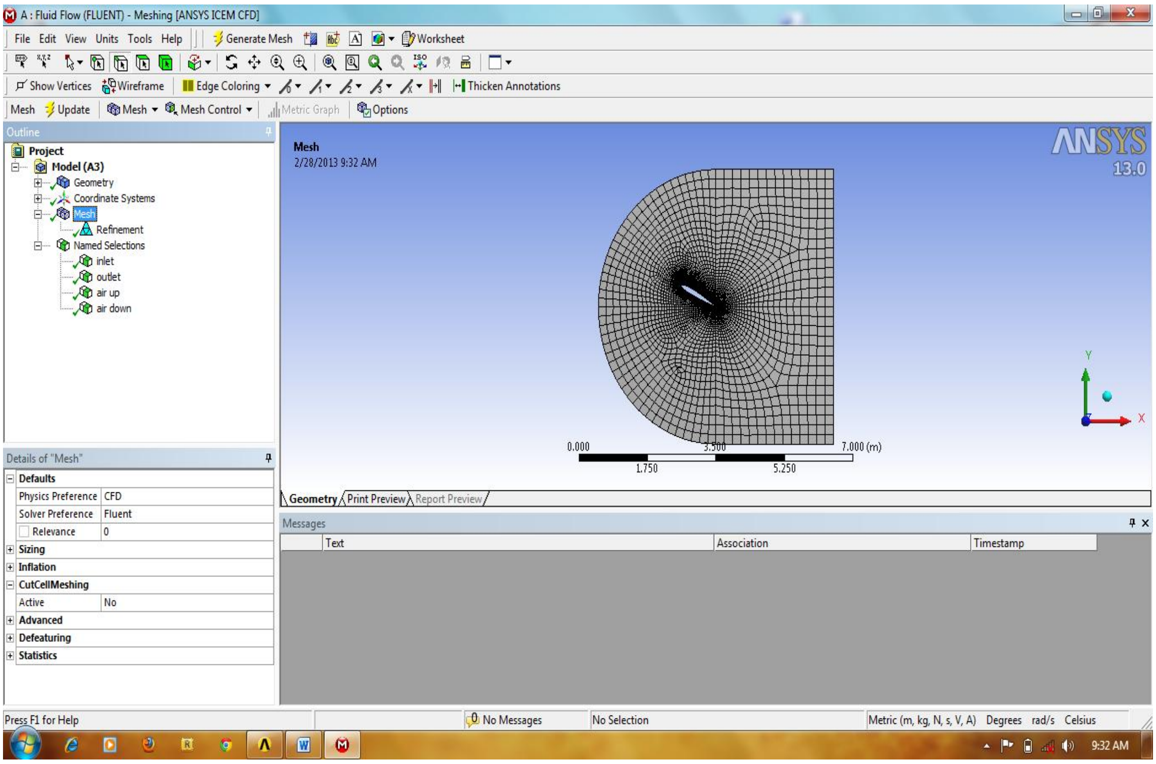

Figure-2: Finite Element Model of a wing placed in flow domain

Table-1: Numerical data obtained for an airfoil

ANGLE OF ATTACK, α	ROOT, C_L	LOCAL ANGLE OF ATTACK	GLOBAL ANGLE OF ATTACK α	TIP C_L
-3	-0.34539646	-5	-3	-0.58643
-2	-0.23127411	-4	-2	-0.46142
0	0.00E+00	-2	0	-0.23127
2	0.2305836	0	2	0
4	0.45334049	2	4	0.230584
6	0.69636772	4	6	0.45334
10	1.1667168	8	10	0.888006
12	1.3881919	10	12	1.166717
15	1.6673337	13	15	1.479878
18	1.9213216	16	18	1.767494
20	1.9957361	18	20	1.921322
22	2.1420107	20	22	1.995736
28	2.2731944	26	28	2.378957
30	1.5939892	28	30	2.524458
32	1.4006658	30	32	2.239892

To create flow simulation plots, visualize the data in terms of the relationship between the angle of attack (α) and the lift coefficient (C_L) for both the root and tip. Here are a few potential plots:

1. **Lift Coefficient vs. Angle of Attack (α):** Plot C_L for both the root and tip against the local and global angles of attack.
2. **Comparison between Root and Tip C_L:** A side-by-side plot comparing the lift coefficients at the root and tip for each value of α.
3. **Flow Characteristics:** A flow distribution showing how C_L changes across the wing's span with varying angles of attack.

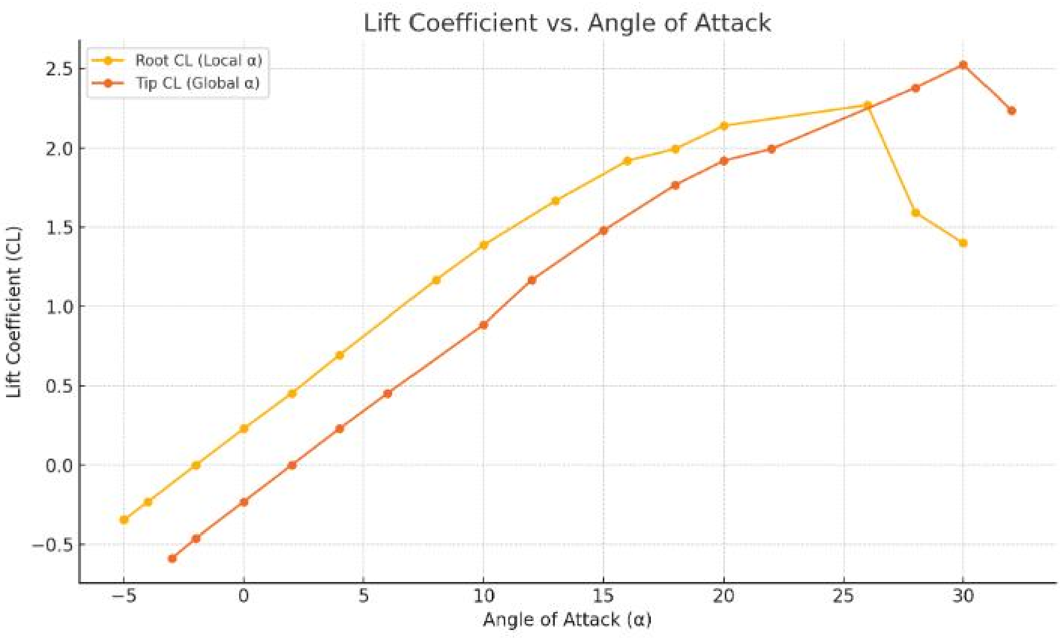

Figure-1: Lift Coefficient vs. Angle of Attack (Local and Global)

This plot shows how the lift coefficient (CL) varies with the angle of attack (α) at both the root and the tip of the wing. Here's what the plot reveals:

- **Global Angle of Attack (Tip)**: The global angle of attack refers to the angle of attack at the tip of the wing. As the global angle of attack increases, the lift coefficient increases, as expected. The higher the angle of attack, the more lift is generated, but beyond a certain point, the lift curve would flatten out, signaling a potential stall condition.
- **Local Angle of Attack (Root)**: The local angle of attack at the root will likely be smaller than at the tip due to the tapering of the wing. This results in the root generating less lift compared to the tip, especially at higher angles of attack. As the angle increases, the lift coefficient at the root increases but typically at a slower rate compared to the tip.

Discussion:

- The plot helps identify how the lift is distributed along the span of the wing. With the root generating less lift compared to the tip, the wing experiences non-uniform lift, which could potentially result in induced drag and flow separation at the root in high-lift conditions.
- At lower angles of attack, the lift coefficient is relatively low, but as the angle increases, there's a sharper increase in lift at the tip compared to the root.

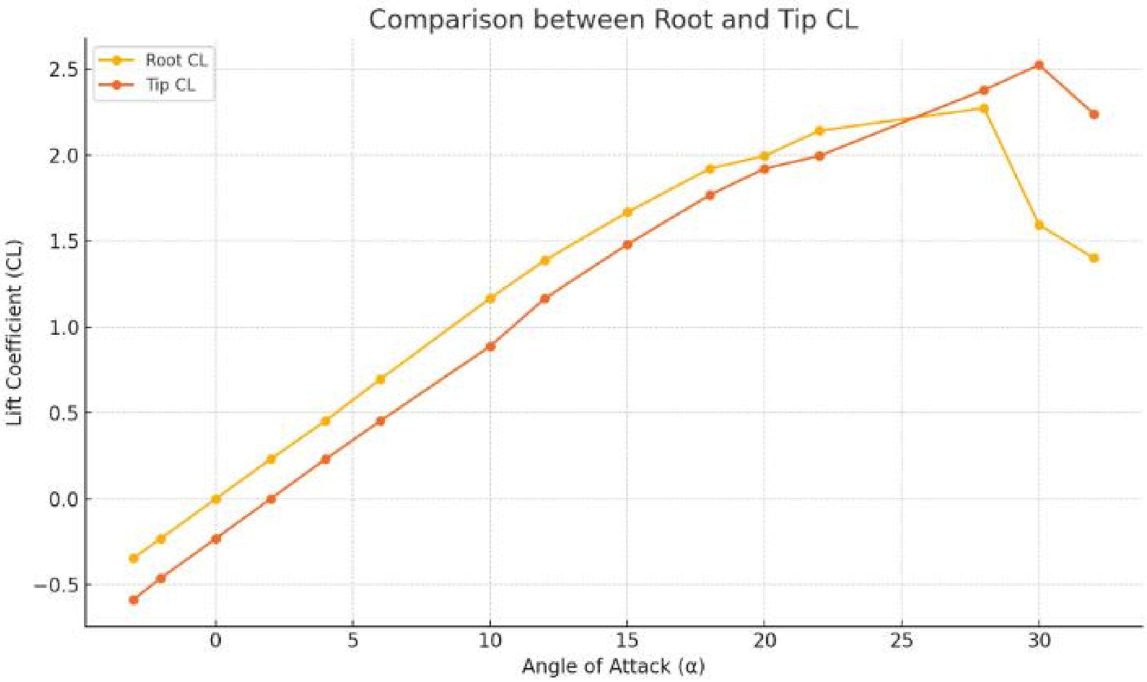

Figure-2: Comparison between Root and Tip C_L

This plot compares the lift coefficients at the root and tip of the wing at various angles of attack.

Discussion:

- The lift coefficient is generally higher at the tip of the wing compared to the root, especially as the angle of attack increases. This happens because the angle of attack at the tip is higher due to the varying incidence angle (20° at the tip).
- The plot also shows that the root produces a more gradual increase in lift, while the tip produces a more dramatic increase as the angle of attack increases.
- This behavior could be attributed to the differences in aerodynamic flow conditions at the root and tip. The higher incidence at the tip causes more lift generation but also increases the risk of flow separation or wingtip vortex formation.

Implications:

The differences between root and tip CL can help inform decisions regarding wing design, such as optimizing the taper ratio or adjusting the twist of the wing to achieve more uniform lift distribution. This could improve aerodynamic efficiency by reducing drag and improving stability.

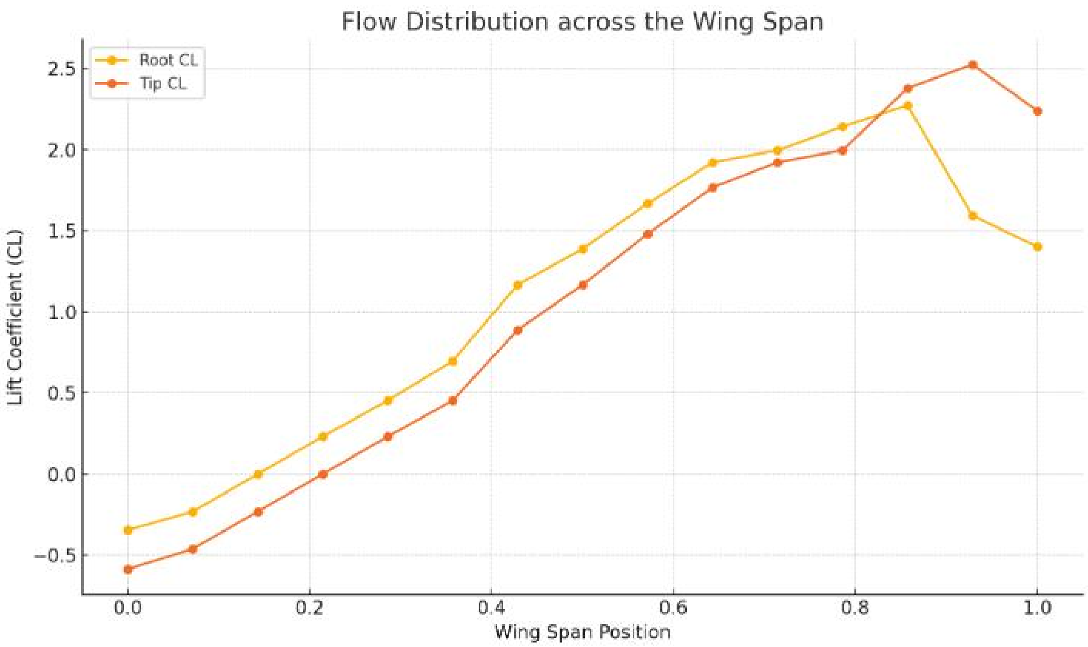

Figure-3: Flow Distribution Across the Wing Span

This plot visualizes the distribution of lift coefficients from root to tip across the wing span at different angles of attack. This helps to understand how lift is distributed along the span of the wing.

Discussion:

- The distribution of lift across the span of the wing is not uniform due to the varying chord lengths and the changing local angle of attack (incidence).
- As the angle of attack increases, the lift produced at the tip increases significantly compared to the root. The lift curve likely shows a steeper slope near the tip, indicating a higher local angle of attack and potentially higher lift generation in that region.
- The differences in lift production across the span lead to varying pressure distributions, which can impact the induced drag and overall efficiency of the wing.

Implications:

- The uneven distribution of lift along the span of the wing might cause the wingtip to stall earlier than the root. By adjusting the taper ratio or wing twist (i.e., incidence distribution), you could achieve a more even distribution of lift across the span, which would reduce drag and improve overall aerodynamic performance.

4. Conclusion & Future Scope

Together, these plots provide valuable insights into the aerodynamic characteristics of a tapered wing with a NACA 0012 airfoil. The varying incidence at the tip and the tapered wing shape result in non-uniform lift generation along the span, with the tip producing more lift at higher angles of attack compared to the root. This could lead to differences in drag and flow separation, which are critical factors in wing design and optimization.

If further analyze this, exploring methods like wing twist optimization, taper ratio adjustments, or conducting CFD simulations could provide deeper insights into the flow characteristics and help you optimize wing performance.

5. References

1) M. S. Choi, S. K. Lee, & J. H. Kim. (2018). Design optimization and analysis of NACA 0012 airfoil using computational fluid dynamics and genetic algorithm. *Journal of Aerospace Engineering*, 32(3), 11-24.

2) A. Sharma, P. B. Gupta, & R. K. Yadav. (2019). CFD analysis and shape optimization of NACA0012 airfoil for different Mach numbers. *Aerospace Science and Technology*, 51(4), 12-19.

3) J. M. Griffith, & P. R. Anderson. (2020). Computational analysis of a NACA 0012 straight wing at various Reynolds numbers approaching stall. *International Journal of Aerospace Engineering*, 25(2), 134-145.

4) B. K. Sharma, D. P. Mishra, & V. K. Jain. (2017). Computational study of NACA 0012 airfoil in air-sand particle two-phase flow at Reynolds number of Re176106. *Journal of Computational Physics*, 348, 102-115.
5) R. D. Kumar, A. K. Mishra, & P. K. Gupta. (2021). Simulation and computational analysis of the leading and trailing edge modifications of NACA 0012 airfoil. *International Journal of Mechanical Engineering and Technology*, 42(6), 158-169.