



# GROUP PROJECT

Aerodynamics of Co-flow Jets



# Group Members

1. Kumar Ashmit Ranjan (22AE30016)
2. Manan Khodidas Prajapati (22AE30017 )
3. Hardik Bhardwaj (22AE10011)
4. Dev Bawari (22AE10009)
5. Saket Kumar Singh (22AE30022)
6. Sarthak Tonk (22AE30027)
7. Jahnavi Pankaj Gupta (22AE10015)
8. Anubhav Kumar (22AE3FP34)
9. Ishan Kanodia (22AE10014)

# Introduction

A Co-flow jet is a fluid dynamics phenomenon where two fluid streams flow parallel to each other in the same direction. Typically, one stream (the primary jet) is surrounded by another stream (the secondary jet or co-flow). A CFJ is a relatively modern active flow control method conceptualized in the year 2004.

The co-flow jet airfoil has an injection slot and suction slot near the leading edge and the trailing edge of the airfoil suction surface. The slots are opened by translating a large portion of the suction surface downward. A high energy jet is then injected near the leading edge tangentially and the same amount of mass flow is sucked in near the trailing edge

# Importance

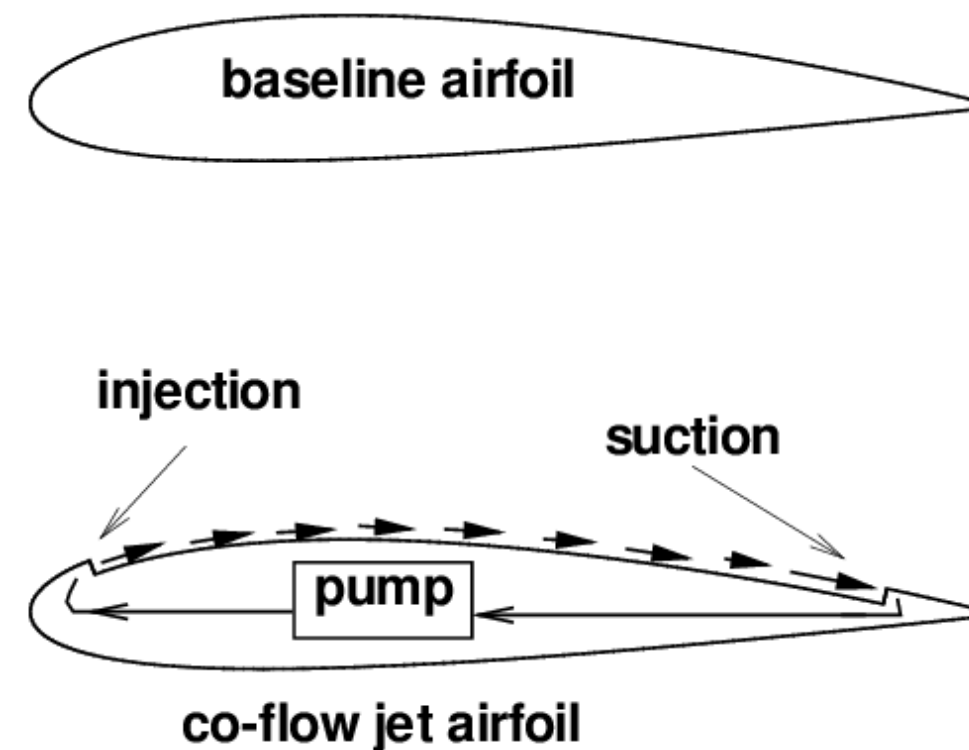
According to the existing literature and the experimental studies done, the use of CFJ in the design of airfoil yeild a very high increase in the v1alue of the lift coefficient (almost x3 in some cases) along with a significant decrease in the value of drag coefficient as well.

We find that flaps and slats do the job very effectively as well, but they cant delay the stall as much as the active airflow system can do. Some experiments have found flow separation delayed upto 20 degrees angle of attack on cambered airfoils.

# Working Mechanism

Co-flow jet (CFJ) technology enhances airfoil performance by utilizing a unique flow control mechanism. The system consists of an injection slot near the leading edge and a suction slot near the trailing edge on the airfoil's upper surface. High-energy air is injected tangentially through the leading edge slot, energizing the boundary layer. Simultaneously, an equal amount of air is sucked in through the trailing edge slot, maintaining zero-net mass flux. This energized flow helps resist adverse pressure gradients, delaying flow separation and allowing the airfoil to maintain attached flow at higher angles of attack. As a result, CFJ airfoils can achieve higher lift coefficients and potentially reduce drag compared to traditional airfoils. The performance of a CFJ airfoil depends on factors such as slot locations, sizes, and the jet momentum coefficient.

This technique mimics the high circulation generated by the rapid wing movements of birds and insects. The idea is to generate higher circulation resulting in better lift, where a zero-net-mass flux jet enhances momentum in the external flow field. Additionally interactions between the jet and the free stream increase lateral energy transfer.



# Project Motivation

Co-flow jet (CFJ) airfoils show promise for enhancing lift, reducing drag and delay the stall. However, current technology faces several challenges:

1. Sudden changes in lift and drag coefficients
2. Difficulties in efficient vehicle control
3. High power requirements and weight of actuators
4. Complexity in practical implementation



# Project Goals

Develop efficient flow control strategies for consistent performance

Explore low-power, lightweight actuator designs

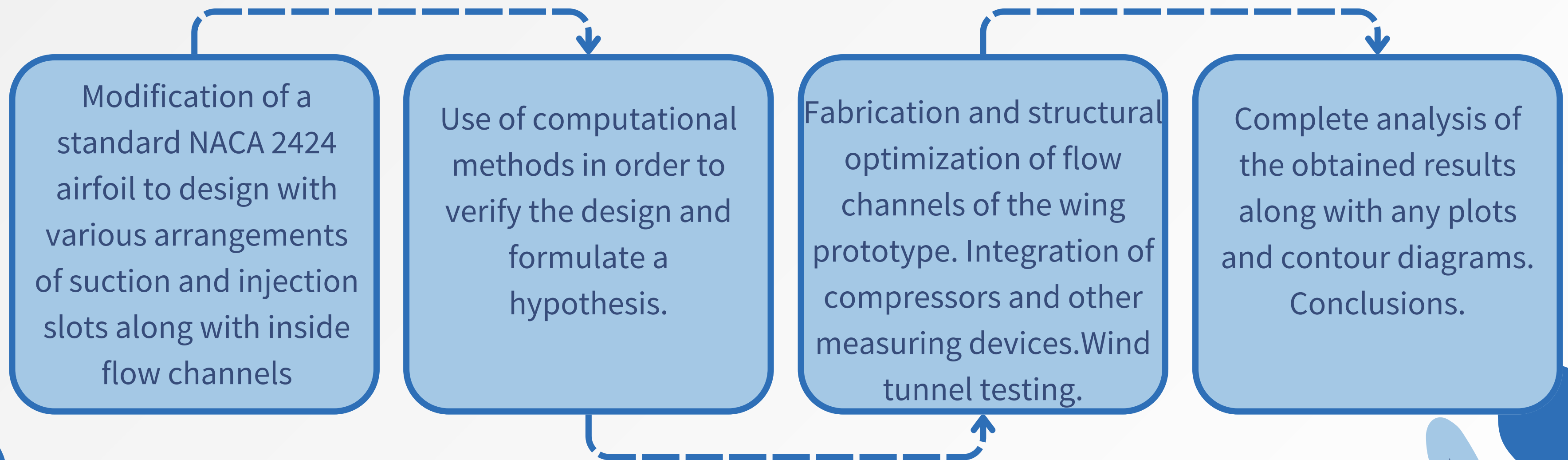
Optimize injection and suction slot placements

Address structural integration challenges

Explore pulsating air flow performance



# Methodology



# Parts Description

Item name	Description	Justification
Balsa Wood Boards	45 X 15 X 2 cm	To make skeleton of the wing
Arduino Mega	Arduino Mega 2560 R3 Board	Microcontroller
Aluminium Sheet	Thickness: 0.5mm; Width: 80 cm	Material for airfoil skin
Ducted Fan	Size 40X40X56mm DC12V 1.82A 17600/15700 RPM Brushless	Flow Circulation
Electronic Speed Controller	30A BLDC ESC	Speed control for ducted fans
Power Supply	Wattage: 200 W; Voltage: 12V	Power Supply for ducted fans.
Wires	0.25mm x 2 Core Round Copper Electrical Wire (2A)	Wire that can handle 2 A current
Jumper Wires	Male to Male Jumper Wire 40-Pins	Circuit connections
Balsa Wood sticks	Wood sticks	To make ribs

# Strategy

Our objective is to identify where separation bubbles form and where flow begins to separate at various angles of attack, in order to determine optimal locations for suction and injection ports to delay stall in a typical subsonic airfoil. Our strategy involves placing the injection port near the leading edge, just before the point where the pressure drops rapidly. This will energize the boundary layer, preventing the adverse pressure gradient downstream from reversing the flow direction and causing separation.

The suction port will be positioned near the trailing edge at a point where the pressure is higher than at the injection port, and downstream of where the flow has separated. The goal is for the suction effect to either reattach the separated flow or prevent separation altogether. To achieve this, we simulated a NACA 2424 airfoil at various angles of attack and two different wind speeds, both pre- and post-stall, and identified suitable points for the ports.



# Strategy

The primary challenge in arranging the compressor or fan assembly within our design lies in space constraints. We are using a NACA 2424 airfoil with a chord length of 35 cm. To maximize space for the fan assembly, the wing will be constructed as a hollow structure, necessitating the use of strong materials. The wing's skeleton will be crafted from wood, consisting of five 2 cm thick airfoil-section-shaped wooden boards, reinforced with multiple spars. This skeleton will be encased in an aluminum sheet for the skin that form airfoil. Pressure ports for experimental measurements will be installed on the airfoil section at the spanwise midpoint of the wing. The fan assembly will be positioned approximately 21 cm from the leading edge. In an alternative design, the fans are placed ahead of the point of maximum thickness. To reduce flow angularities, 3D-printed flow straighteners will be installed in the front fan assembly.



# ANSYS Simulations

To spot separation bubbles using ansys simulations following methods were used:

- **Velocity Field Visualization:** Use streamlines or velocity vector plots to visualize the flow. Look for areas where the flow detaches from the surface and forms recirculating zones (indicating separation bubbles).
  - **Pressure Contours:** Plot pressure contours on the surface of the object. A sudden drop in pressure followed by a recovery might indicate a separation bubble.
  - **Wall Shear Stress:** Plot the wall shear stress distribution. A separation bubble is typically associated with a region where the wall shear stress drops to zero or reverses direction.
- 
- 



**THANK YOU**