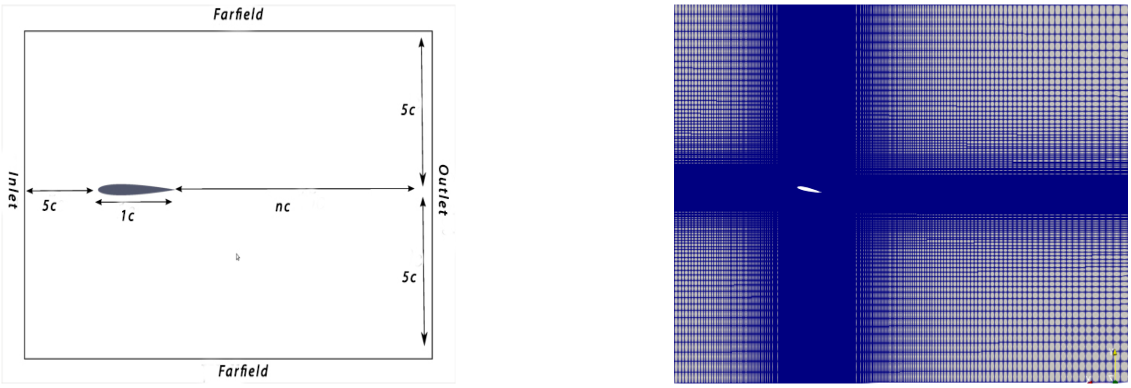
Flow Prediction around a NACA0012 Airfoil at low Reynolds Number

Subham Kant Das

Abstract

Since the past few decades, there has been an increased interest in the field of Micro Aerial Vehicles(MAVs) and their potential application in the defense sector. Their reduced size, coupled with higher payload capacity, has extended their use to various other sectors. However, due to the smaller size followed by shorter wingspan, the chord-based Reynolds Number for MAV is very less(of the order of 1000), which leads to poor performance in the range of operation. Hence to improve their performance, it becomes essential to study the flow characteristics at such low Reynolds Number. The flow in this regime consists of complex phenomena such as the transition from laminar to the turbulent regime, flow separation, and reattachment followed by the formation of laminar separation bubble. Although the flow is initially laminar, the entire phenomenon cannot be captured using conventional laminar models. Thus the flow around airfoils at low Reynolds Number is intriguing to researchers.



(a) C-domain **(b)** O-domain

Figure 1: Computational Domain to investigate the effect of boundary proximity and mesh resolution on numerical results.

The objective of this study is to numerically examine the effect of solver flow type(laminar or turbulent), meshing resolution and domain extent on the flow physics around a NACA0012 Airfoil at low Reynolds Number(1000). The rectangular domain shown in Figure 1(a) is used for the present study since the mesh can be easily controlled near the Trailing Edge to capture the wake. The present work will evaluate the ability of the laminar and Spalart Allmaras(SA) model to predict the flow behavior at a Reynolds Number of 1000 using the PISO algorithm in OpenFOAM. The effect of domain extent and mesh resolution will be studied, and the results will be validated against available literature.