CFD analysis of a Symmetrical Airfoil with Influence of Reynolds Number and Attack Angle

Abstract:

A Detailed Study and Computational Fluid Dynamics investigation was conducted to ascertain and highlight the trends over a symmetrical airfoil under the effect of the Reynolds number and angle of attack.The analysis was carried out by varying the angle of attack from 0^0 to 10^0.It was found that high values of pressure coefficient are obtained on the lower surface. This region of high pressure extended almost over the entire lower surface for higher angles of attack.The pressure distribution on the upper surface did not change significantly for higher angles of attack.The results showed that the streamwise variations of the pressure coefficients of the airfoil showed similar distribution at both Reynolds numbers, while the stall angles and lift coefficients increased with Reynolds number. Further the results were validated with the experimental data obtained from literature.

Problem Statement:

Analysis the air flow over NACA 0012 airfoil at from 0^0 and 10^0 angle of attack. Use our visicd model. At the inlet of the system, we will define the velocity as entering at a 0^0 to 10^0 angle of attack (as per the problem statement), and at a total magnitude of 1. We will also define a gauge pressure at the inlet to be 0. As for the outlet, the only thing we can assume is that the gauge pressure is 0. As for the airfoil itself, we will treat it like a wall. Calculate the lift and drag coefficient for unrefined and refined mesh and compare it with experimental data.(use it from internet)

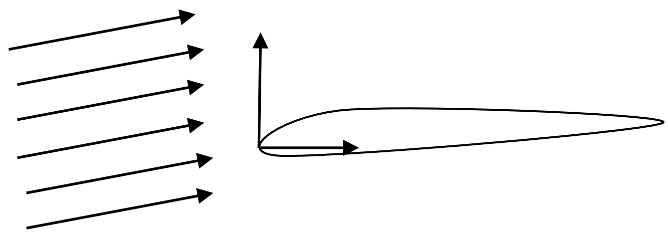

Fig 1.0

Following are the initial condition that you will need to solve the problems.

Inlet : x-velocity = 0.9945m/s

y-velocity = 0.1045m/s

Gauge pressure = 0

Outlet: Gauge pressure = 0

To validate our data, we can compare values to actual experiment, the drag coefficient coming from our inviscid model, however, cannot be compared to actual data. The drag coefficient in our model is theoretically zero.