**Computation of Lifting Flow about a Flat Plate**

The purpose of the simulation is to obtain a numeric solutions for the **Laplace equation** for the velocity potential. It is given that the fluid is incompressible and that the flow is inviscid and irrotational. By imposing the given **steady boundary conditions**, a lift is generated which is then computed.

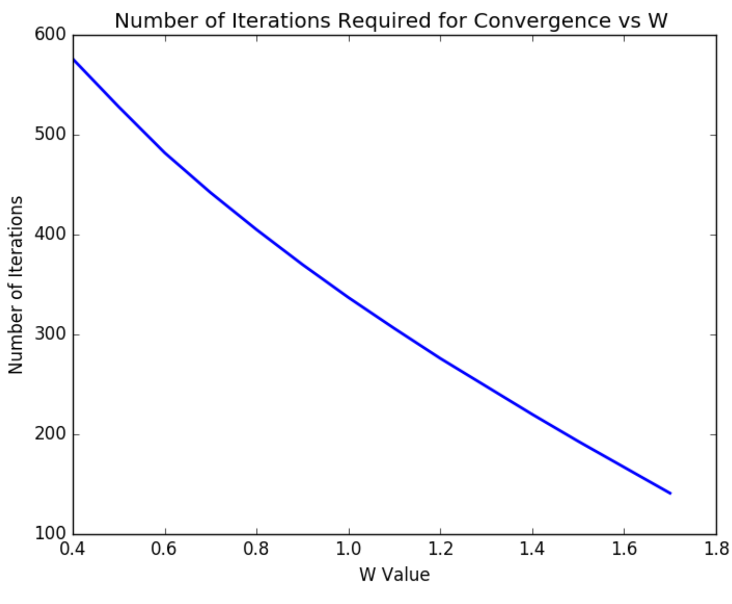
First, the grid layout was constructed with equations Z1-Z9. Then, the difference and update equations (NN1-NN23) for the applications of various boundary conditions were used to update the $\Phi$ matrix.

When updating the equations, the *w* values used gave different number of iterations required for convergence. For $w \in [0.4, 1.7]$, the number of iterations required for convergence for recorded.

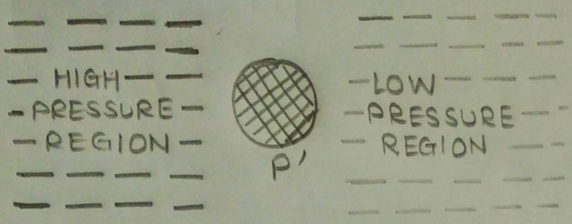
It was observed that increasing the *w* value decreased the number of iterations required for convergence, which is in sync with the hypothesis that a larger *w* value increases the speed of the computation.

For team-specific *l*[*i*] and ε=0.001

| w Value | Number of Iterations |
|---|---|
| 1.7 | 141 |
| 1.6 | 167 |
| 1.5 | 193 |
| 1.4 | 220 |
| 1.3 | 248 |
| 1.2 | 276 |
| 1.1 | 306 |
| 1.0 | 337 |
| 0.9 | 370 |
| 0.8 | 405 |
| 0.7 | 442 |
| 0.6 | 483 |
| 0.5 | 528 |
| 0.4 | 576 |



Consider a particle P':




Force (F) acting on particle P' = $-\nabla P \cdot (\text{Vol. of } P')$

$$F = m \frac{d\vec{V}_{P'}}{dt} \quad [\text{Newtons' II}^{nd} \text{ Law}]$$

$$\therefore \rho_{P'} (\text{Vol. of } P') \frac{d\vec{V}_{P'}}{dt} = -\nabla P \cdot (\text{Vol. of } P')$$

$$\Rightarrow -\nabla P = \rho\left(\frac{\partial \vec{V}_{P'}}{\partial t} + V_{P'} \nabla V_{P'}\right)$$

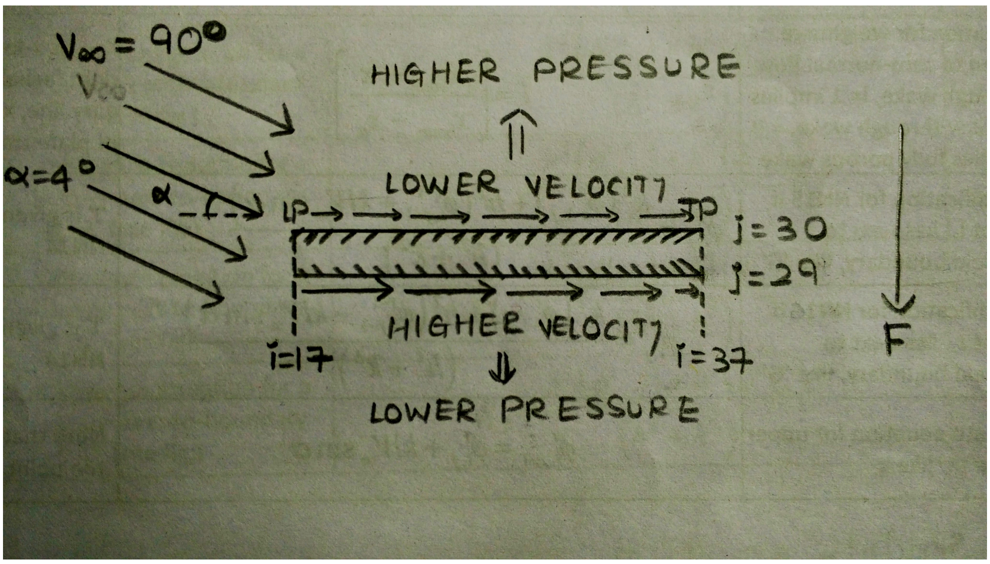
Where $\frac{dV}{dt} = \frac{\partial V}{\partial t} + V \cdot \nabla V$, material derivative has been used.

$$\boxed{\therefore \frac{\partial \vec{V}}{\partial t} + V \cdot \nabla V = -\frac{1}{\rho} \cdot \nabla P}$$ Eulers' Equation

It can be observed that if velocity is higher, pressure is lower. [and vice versa]

The lift on an aerofoil is primarily the result of its angle of attack and shape. When oriented at a suitable angle, the airfoil deflects the oncoming air, resulting in a force on the airfoil in the direction opposite to the deflection. This force is known as aerodynamic force, which has two components. The component of this force perpendicular to the direction of motion is called lift. The component parallel to the direction of motion is called drag.
This "turning" of the air in the vicinity of the airfoil creates curved streamlines, resulting in lower pressure on one side and higher pressure on the other. This pressure difference is accompanied by a velocity difference, via **Bernoulli's principle**, so the resulting flow field about the airfoil has a higher average velocity on the upper surface than on the lower surface.

For this simulation, we have adopted the **thin airfoil theory**. This theory relates the angle of attack to lift for incompressible, inviscid flows. An idealisation which is used is that the airfoil can be imagined to be of zero thickness and infinite wingspan.
According to this theory,
***Lift Coefficient:***
$c_l = 2\pi\alpha$




But in our simulation, the plate has a finite length. This violation of the thin airfoil theory by itself will give rise to some error.

**Bibliography:** *https://en.wikipedia.org/wiki/Airfoil*
https://www.youtube.com/user/stevemilwa

**B3-Group E**
**Divyam Khandelwal 15056**