

# CFD analysis and wind tunnel experimental validation of NACA4412 airfoil

Pravin Katare, Shantanu Sapre

<sup>1,2</sup>Department of Mechanical Engineering  
Marathwada Mitra Mandal's College of Engineering,  
Behind Vandevi Temple, Karvenagar, Pune- 411052

## ABSTRACT

Computational methods have been used in the field of Aerodynamics as innovative tool. The investing parameters associated with the aircraft such as force coefficients, critical angle of attack, lift and drag forces, etc. can be predicted by computational methods. The study focuses on 2D analysis of flow over NACA4412 airfoil to optimise turbulence model, boundary conditions and validating the results with experimentation in a wind tunnel. The airfoil is investigated by generating a different turbulence models with variation of angle of attack. The result obtained by conducting experiments in wind tunnel and analysing it using CFD tool will be useful for further development in aeronautical industry.

Keywords

CFD , Wind Tunnel, validation, NACA4412 airfoil

## ABBREVIATIONS

<u>Symbol</u>	<u>Significance</u>
$v$	Velocity of free stream
$\mu$	Dynamic viscosity of air
$\alpha$	Angle of Attack (AOA)
$Re$	Reynolds Number
$\rho$	Density of fluid
$C_l$	Coefficient of Lift
$C_d$	Coefficient of Drag
$C$	Chord of Airfoil

## INTRODUCTION

Computational fluid dynamics is a method of analysing and solving the problems involving fluid flows by using numerical methods. Evolution of high speed computers made it possible to solve many complex engineering problems and get fairly accurate results. These methods were primarily used in the aerospace industry for simulation of airflow over the aircraft and save the time involved in prototyping and testing. The CFD analysis involves 2D as well as 3D approach. The analysis of the object using CFD may be validated through wind tunnel testing to obtain the technical solutions for complex fluid. Any complex wing analysis is carried out in 2D on an airfoil and extended it with 3D wing. However, validation of the CFD results is important to avoid the errors in CAD, meshing quality, selection of turbulence model etc. The geometric model or prototype is fabricated and tested inside a wind tunnel. The CFD results are compared with the wind tunnel testing. The simulating results developed a airfoil objects for the industrial application.

## METHODOLOGY

Relevant research papers and forums related to CFD and wind tunnels were reviewed for the development of airfoil. CFD solver can generate different result with variation of investigating parameters for each physics model. The selection of appropriate model is very essential to get accurate results. It was done by taking into account, the methodology and findings from the literatures referred.

A CFD analysis was carried out on NACA4412 for obtaining the lift and drag coefficients and tested it inside a wind tunnel. This is a very commonly used airfoil in many applications such as aircraft wings and turbine blades as it provides sufficient lift at low Reynolds number <sup>[1]</sup>.

The work is completed considering the steps shown in figure 1.

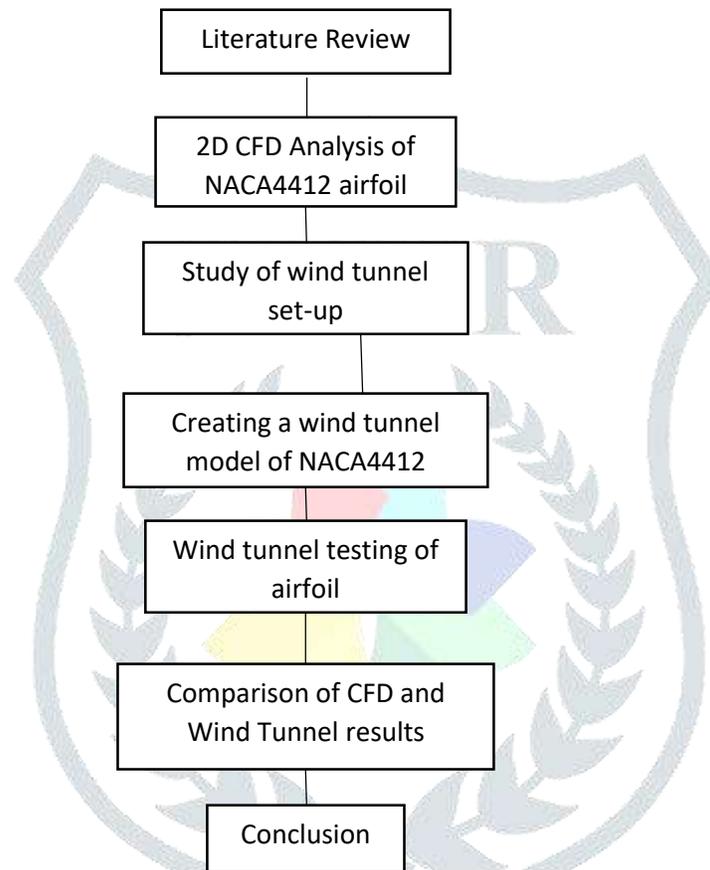


Figure 1: Methodology Flow Chart

## CFD ANALYSIS of NACA4412 Airfoil:

The 2-D CFD analysis of NACA4412 Airfoil carried out using Ansys Fluent in sequential steps shown in figure 2.

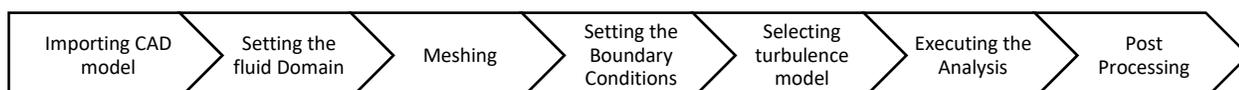


Figure 2: Steps followed in CFD Analysis

Whenever a body moves in a fluid with a considerable speed it creates a disturbance in the

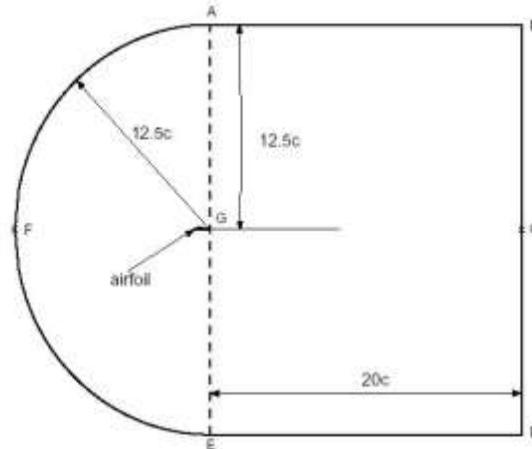


Figure 3: Airfoil fluid Domain [2]

surrounding region over a significant distance around it. For a analysis this fluid domain is created in a following manner where 'C' represents the Chord of Airfoil as shown in figure 3.

Meshing plays an important role in this analysis. To obtain fairly accurate results, the meshing closer to the airfoil boundary should be very fine and structured as shown in the figure 4 and figure 5.

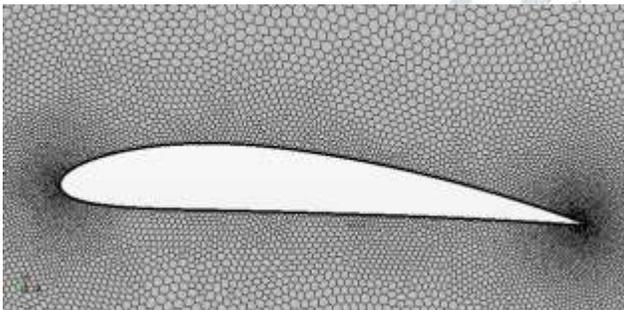


Figure 4: Meshing around airfoil

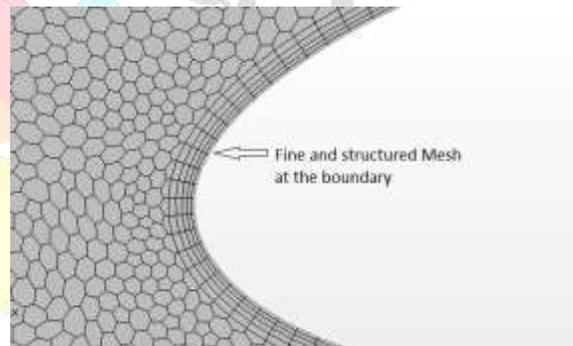


Figure 5: Structured mesh at the boundary

#### Parameters:

Chord (C): 0.15 m

Linear Velocity (v): 15 m/s

Range of Angle of Attack ( $\alpha$ ): 0 – 16°

Density of Air ( $\rho$ ): 1.177 kg/m<sup>3</sup>

Dynamic viscosity of air ( $\mu$ ): 1.85 × 10<sup>-6</sup> Pa-s

Reynolds Number (Re):  $\frac{\rho v L}{\mu} = 143,148$

Physics models define the spatial and temporal solution methods and physical properties of the flow. In this case, the flow is steady, turbulent, and compressible. The Spalart - Allmaras turbulence model and the ideal gas model was used. Coupled solver is recommended for such types of flows<sup>[3][4]</sup>

The angle of attack was increased from 0° to 16° at an interval of 2° and the variation in the lift and drag coefficients was studied. The results are represented by velocity contours. Exact values are listed in a tabular format and graphs have been plotted.

**Velocity and Pressure Contours over Airfoil:**

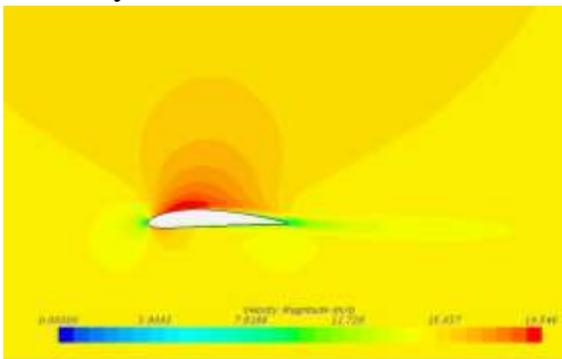


Figure 4: Velocity contours at angle of attack = 0°

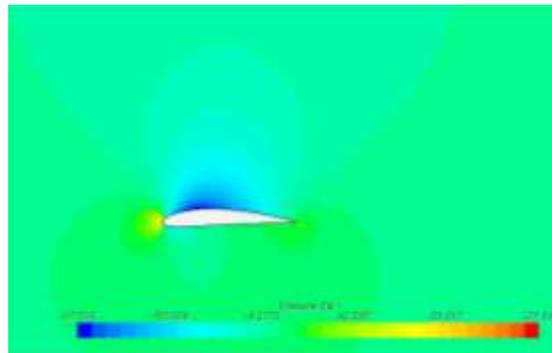


Figure 5: Pressure contours at angle of attack = 0°

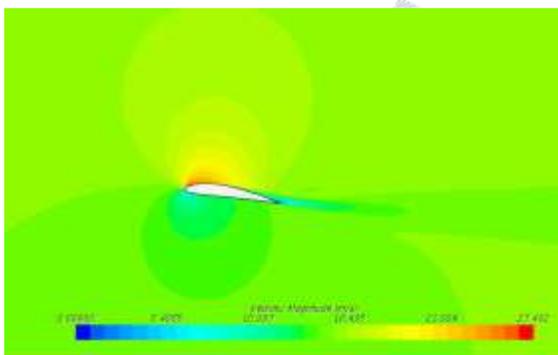


Figure 6: Velocity contours at angle of attack = 8°

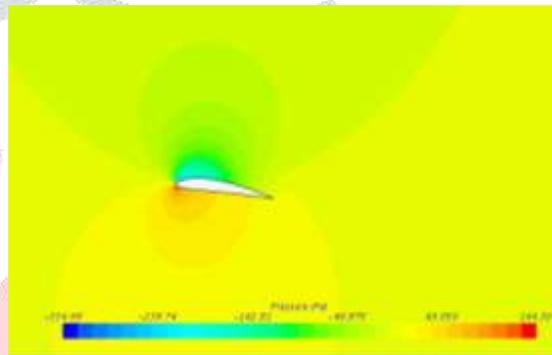


Figure 7: Pressure contours at angle of attack = 8°

Cl/Cd maximum is near 8 angle of attack for this particular Reynolds Number. About this angle of attack, the airfoil will give maximum efficiency for this Reynolds number

The velocity and pressure plots indicate that the airflow over the airfoil has become turbulent. A partial

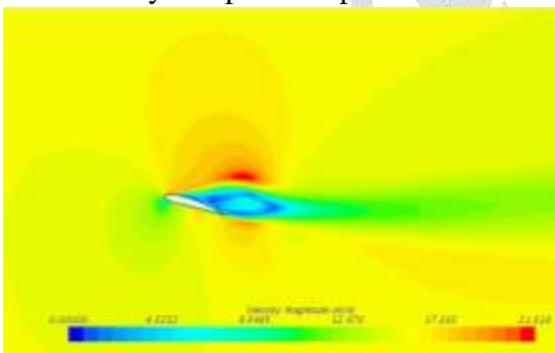


Figure 8: Velocity contours at angle of attack = 16°

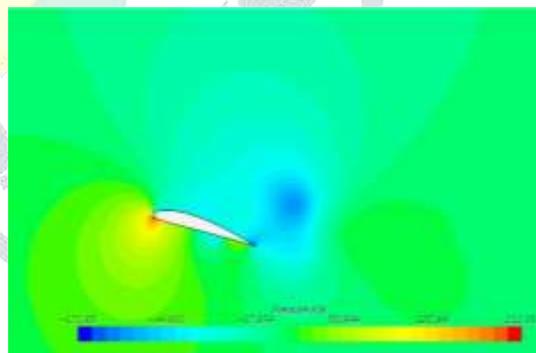


Figure 9: Pressure contours at angle of attack = 16°

vacuum is created in the bluish region signifying that the separation of airflow occurs ahead of the trailing edge and the airfoil stalls.

Results:

Angle of Attack ( $\alpha^\circ$ )	Cl	Cd	Cl/Cd
0	0.4294	0.01967	21.8302
2	0.6223	0.02083	29.87518
4	0.8507	0.02236	38.04562
6	1.0245	0.02297	44.60165
8	1.2240	0.02402	50.95754
10	1.3523	0.03204	42.20662
12	1.4501	0.04316	33.59592
14	1.5002	0.0804	18.65920
16	1.2012	0.2224	5.395683

Table 1: CFD Results

Observations:

It was observed that as the angle of attack increases, there is an increase in the coefficient of lift as well as the coefficient of drag. The Cl/Cd denotes the efficiency of the airfoil. For this particular Reynolds number the maximum efficiency is obtained between  $6^\circ - 8^\circ$  angle of attack. After this, the flow separation starts occurring ahead of the trailing edge, hence there is a decrease in the efficiency of the airfoil. Further increment in the angle of attack leads to flow separation almost at the maximum camber of the airfoil and there is a sudden drop in the lift. This is the stalling point of the airfoil which is observed around  $16^\circ$ .

## WIND TUNNEL SETUP

Wind tunnels are primarily classified into two categories – Open Return Wind Tunnel and Closed Return Wind Tunnel. Most of the wind tunnels consist of the following parts: inlet, convergent section, test section, diffuser, and fan /airflow generator. A honeycomb mesh is introduced in the inlet in order to generate linear flow inside the test section. The wind tunnel used for this study is an open return, low speed wind tunnel. The specifications of this wind tunnel are listed below along with the image.

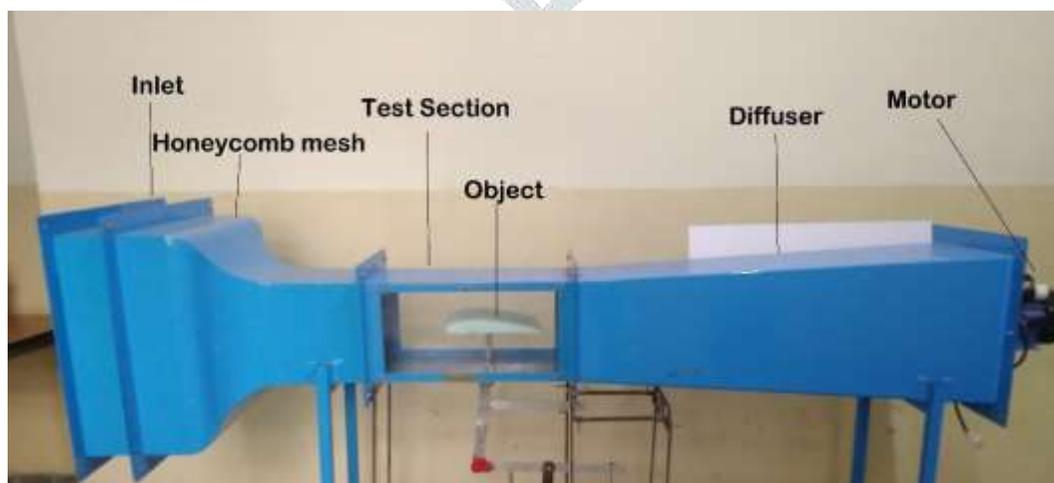


Figure 4: Wind Tunnel Setup

Test Section Dimensions	0.46m × 0.23m × 0.23m
Inlet Dimensions	0.45m × 0.45m
Convergent section length	0.6m
Diffuser length	0.9m
Speed range	5 – 60 m/s
Measurable	Lift and Drag coefficients

Table 2: Specifications of Wind Tunnel

## WIND TUNNEL TESTING

### Physical model:

An airfoil section was cut out from foam with chord 0.15m. Further analysis was performed on this airfoil section.



Figure 5: NACA4412 section

**Testing Procedure:**

The airfoil was placed inside the test section and set at an angle of attack. The airflow was so adjusted that a velocity of about 15m/s was achieved inside the test section. The  $C_l$  and  $C_d$  values were recorded. The same procedure was repeated by increasing the angle of attack as done during the CFD analysis.

Angle of Attack ( $\alpha^\circ$ )	$C_l$	$C_d$	$C_l/C_d$
0	0.13717	0.00000	0.13717
2	0.76419	0.00000	0.76419
4	0.97925	0.00000	0.97925
6	1.52381	0.00000	1.52381
8	1.60727	0.00000	1.60727
10	1.42708	0.00000	1.42708
12	1.04457	0.00000	1.04457
14	0.30304	0.00000	0.30304
16	0.386842	0.00000	0.386842

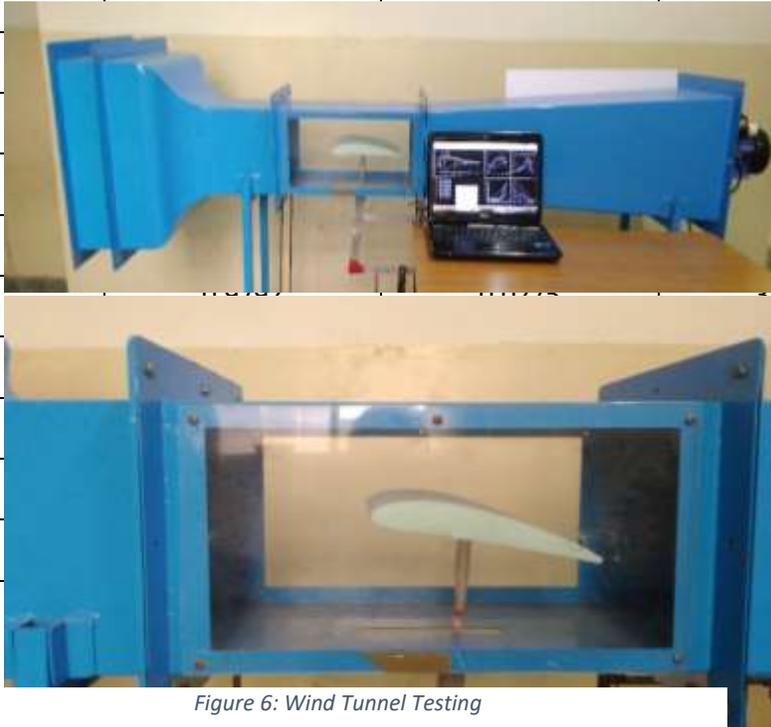


Figure 6: Wind Tunnel Testing

**Results:**

**COMPARISON OF CFD AND WIND TUNNEL RESULTS**

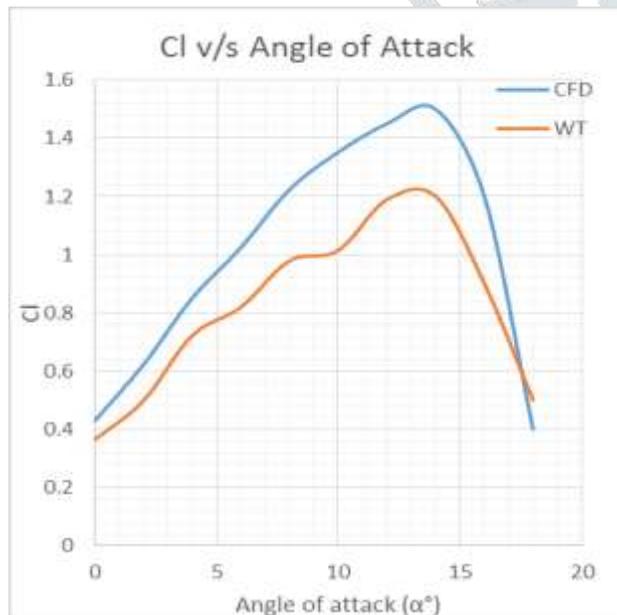


Figure 7: Graph of  $C_l$  vs  $\alpha$

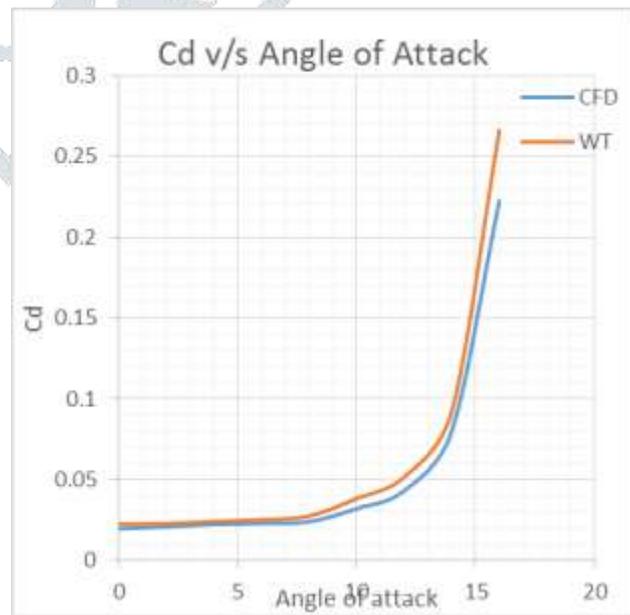


Figure 8: Graph of  $C_d$  vs  $\alpha$

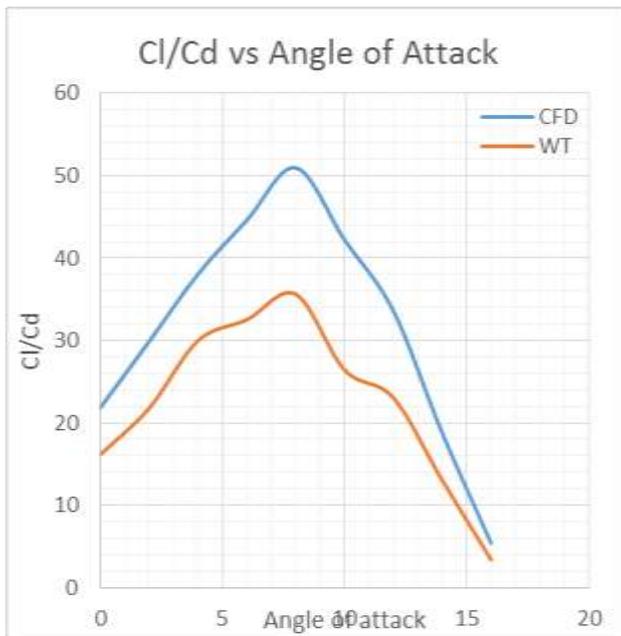


Figure 9: Graph of Cl/Cd vs  $\alpha$

- The Cl observed from the CFD is greater than that observed in the wind tunnel
- The Cd observed is more in wind tunnel than the CFD
- Ratio Cl/Cd is significantly reduced for the wind tunnel testing than the CFD results

## CONCLUSION

CFD analysis gives us a fair idea and solutions for the problem, however, wind tunnel testing provides more realistic and reliable solutions. In practice, the lift coefficient is less and the coefficient of drag is more than the one calculated from 2D CFD analysis. The reason for above observations is that for 2D CFD analysis, the length of the wing is considered to be infinite. Hence, the wing tip vortices and the induced drag due to it is not considered. While in the case of wind tunnel testing, the wing tip vortices are present and the tip region of the wing does not contribute in the generation of the lift as shown in the figure.

Hence, we get reduced lift and an increased drag in practical than the 2D CFD results. The anomalies in the 2D CFD analysis can be overcome to a considerable extent by performing 3D CFD analysis, however, it requires powerful computers and is a time consuming task.

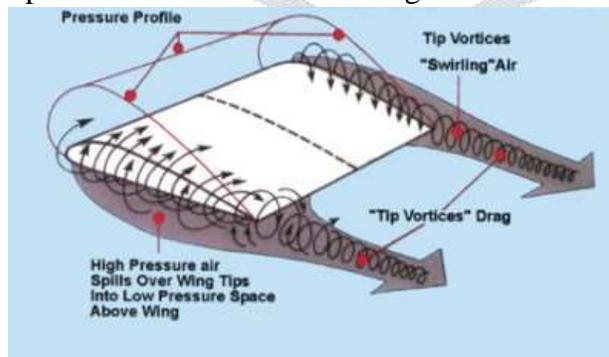


Figure 10: Wing Tip Vortices <sup>[10]</sup>

## REFERENCES

1. Alex E. Ockfen, Konstantin I. Matveev, "Aerodynamic characteristics of NACA 4412 airfoil section with flap in extreme ground effect", International Journal of Naval Architecture and Ocean Engineering, Volume 1, Issue 1, September 2009, Pages 1-12

2. Abas Abdoli, "Simulation of body force field effects on airfoil separation control and optimization of plasma actuator", Journal of Physics D Applied Physics, August 2008
3. S. M. A. Aftab, A. S. Mohd Rafie, N. A. Razak, K. A. Ahmad," Turbulence Model Selection for Low Reynolds Number Flows", April 2016, PLOS One
4. C.D. Argyropoulos, N.C. Markatos," Recent advances on the numerical modelling of turbulent flows", July 2014, Elsevier
5. Mayurkumar kevadiya, Hemish A. Vaidya," 2D ANALYSIS OF NACA 4412 AIRFOIL", International Journal of Innovative Research in Science, Engineering and Technology Vol. 2, Issue 5, May 2013
6. Roy GEBBINK, Ganglin WANG, Min ZHONG," High-speed wind tunnel test of the CAE aerodynamic validation model", Chinese Journal of Aeronautics, (2018).
7. Amit Kumar Saraf, Mahendra Pratap Singh, Tej Singh Chouhan," AERODYNAMIC ANALYSIS OF NACA0012 AIRFOIL USING CFD", International Journal of Mechanical And Production Engineering, ISSN(p): 2320-2092, ISSN(e): 2321-2071 Volume- 5, Issue-12, Dec.-2017, <http://iraj.in>
8. Jasminder Singh, Dr. Jaswinder Singh, Ampritpal Singh , Abhishek Rana, Ajay Dahiya," Study of NACA 4412 and Selig 1223 Airfoils Through Computational Fluid Dynamics", SSRG International Journal of Mechanical Engineering ( SSRG – IJME ) – Volume 2 Issue 6–June 2015
9. <http://airfoiltools.com/airfoil/details?airfoil=naca4412-il>
10. [http://www.pilotfriend.com/training/flight\\_training/aero/wng\\_vort.htm](http://www.pilotfriend.com/training/flight_training/aero/wng_vort.htm)

