



University of Tehran
College of Engineering
School of Mechanical Engineering
Fluid Mechanics II Course



Fluid Mechanics Project

Airfoil Analysis: NACA-2418

Professor:

Dr. Azadeh Jafari

Author:

Mohammad | Montazeri | 810699269

Deadline:

2023/6/30



Fluid Mechanics Project



Index of Contents

Abstract	3
Steps and Processes	4
Geometry	4
Mesh	4
Calculations of Y^+	5
Setup	8
Angle of Attack	12
Results and Analysis	14
Individual Plots	14
Overall Plot	19
Lift and Drag Coefficients and Their Errors	20
Researches	22
a) Prism Layer Meshing and Y^+	22
b) Airfoil Manufacturing Techniques	23
c) Airfoil Simulation Software	23
Appendix	24
References	25



Index of Figures

Figure 1: Geometry of system.....	4
Figure 2: Mesh overview	5
Figure 3: Mesh around the airfoil	5
Figure 4: Experimental diagrams for NACA 2418.....	6
Figure 5: First layer thickness	7
Figure 6: First layer thickness in our mesh.....	7
Figure 7: Mesh quality factors	8
Figure 8: Named selections.....	8
Figure 9: Overview of B.C.s and an example of adjusting inlet B.C.	9
Figure 10: Defining air as the fluid material of system	9
Figure 11: Residuals and equations	10
Figure 12: Operating conditions	10
Figure 13: Defining reports.....	11
Figure 14: The convergence history for the governing equations	12
Figure 15: The effect of angle of attack.....	13
Figure 16: Modification of reports for 3° angle of attack	13
Figure 17: Pressure around airfoil based on its length for 7 different attack angles	19
Figure 18: Drag coeff. based on Lift coeff.	21
Figure 19: Lift coeff based on attack angle	21
Figure 20: Drag coeff based on attack angle	22

Index of Tables

Table 1: Boundary conditions	9
Table 2: Results for 3° angle of attack.....	14
Table 3: Results for 6° angle of attack.....	15
Table 4: Results for 9° angle of attack.....	15
Table 5: Results for 12° angle of attack.....	16
Table 6: Results for 15° angle of attack.....	17
Table 7: results for 18° angle of attack	18
Table 8: Numerical and Experimental Lift and Drag coefficients.....	20
Table 9: Properties of air at atmospheric pressure.....	24



Abstract

An airfoil is a shaped surface that produces lift and drag when moved through the air. Airfoils are designed to manipulate the flow of a fluid to produce a reaction, which in an aircraft's case, is aerodynamic lift. Airfoils are highly-efficient lifting shapes, able to generate more lift than similarly sized flat plates of the same area, and able to generate lift with significantly less drag. Airfoils are used in the design of aircraft, propellers, rotor blades, wind turbines and other applications of aeronautical engineering.

The wings of fixed-wing aircraft feature airfoil-shaped cross-sections. The cross-sectional shape of an airfoil is defined by its thickness-to-chord ratio (t/c) and its camber (the curvature of the upper surface). The thickness-to-chord ratio is the maximum thickness of the airfoil divided by its chord length (the distance from the leading edge to the trailing edge). The camber is the maximum distance between the chord line (a straight line drawn from the leading edge to the trailing edge) and the upper surface of the airfoil.

Airfoils are used in the design of aircraft, propellers, rotor blades, wind turbines and other applications of aeronautical engineering. In airplanes, the lift is the force that opposes the weight of the airplane and is created by the forward motion. When the air passes the wings, it is forced to split between the above and below of the wing. The airfoil shape of the wing causes the air moving over the top surface to move faster than the air moving over the bottom surface. This creates a lower pressure area on top of the wing and a higher pressure area on the bottom of the wing. The difference in pressure creates lift.

Airfoils are also used in propellers and rotor blades. The shape of an airfoil allows it to generate lift when it is rotated through a fluid such as air or water. This lift can be used to power a vehicle or machine.

Wind turbines use airfoils to capture energy from wind. The blades of a wind turbine are shaped like airfoils and are designed to capture as much energy from the wind as possible. As wind passes over the blades, it creates lift which turns the blades and generates electricity.

NACA airfoil sections are a series of airfoils developed by the National Advisory Committee for Aeronautics (NACA). They are created from a camber line and a thickness distribution plotted perpendicular to the camber line. The airfoils are identified by a four-digit number that represents their geometric properties, such as the maximum camber position and the maximum thickness.

The NACA number specified to the author was **23017**. However, since the experimental diagrams of this airfoil was inaccessible in the references, it was altered to **NACA 2418** according to the T.A.'s remark.



Steps and Processes

Geometry

The first step of this project was modeling the airfoil and designing it in ANSYS geometry. The 2D model of the system was based on a list of points that was available in websites. In this model all of the points' coordinates were given. After saving this file and importing it in Design Modeler environment, the domain of the system was sketched. The upper and lower walls of the system were dimensioned 10 times the chord length of the airfoil, which is 1 meter long. The outlet of the system has the same dimension as the walls. Then the airfoil body was subtracted from the domain area. Finally, for better meshing accuracy, the domain got divided into 6 zones, using the *project* command.

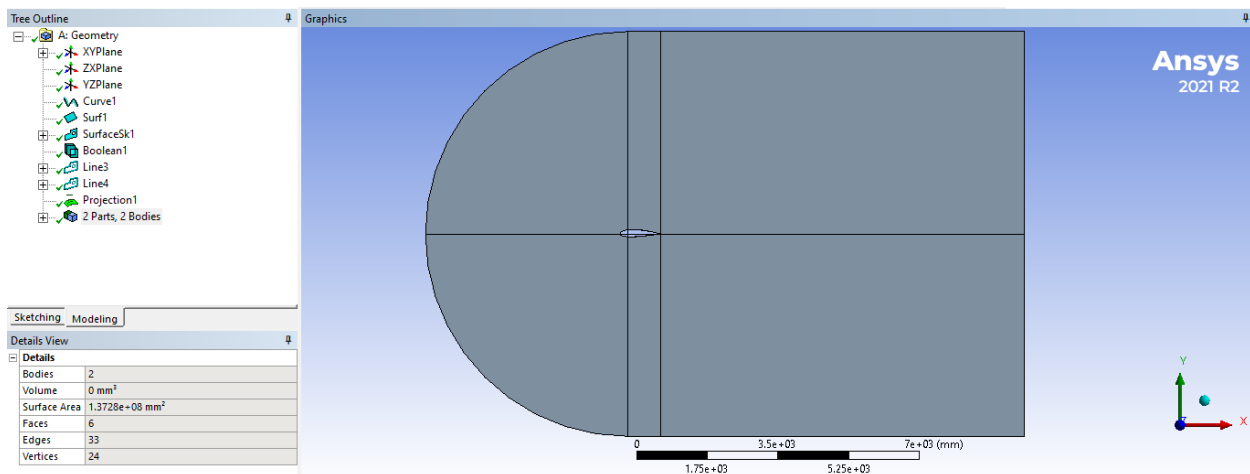


Figure 1: Geometry of system

Mesh

After this geometry had been designed in the Design-Modeler environment using simple circle and rectangle commands, it was imported into Meshing environment. In the meshing procedure, at first an automatic mesh was generated by *Quadrilateral Dominant* method. Then, by inserting *mesh size* commands, the size of boundary elements near airfoil was scaled down to help capturing the boundary layers. This procedure was repeated multiple times for all of the edges. A *face meshing* command was also used to organize the mesh in the best way.



Fluid Mechanics Project

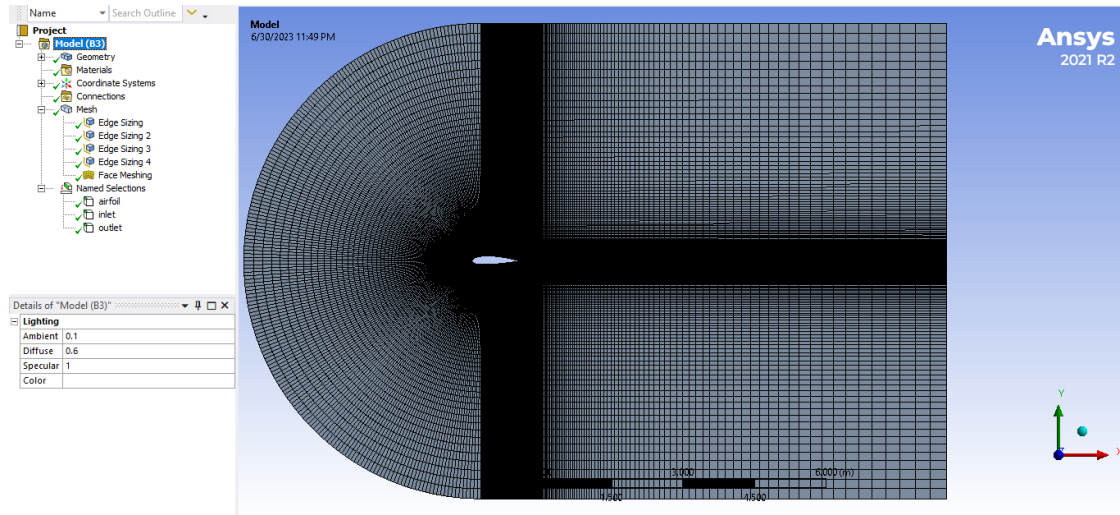


Figure 2: Mesh overview

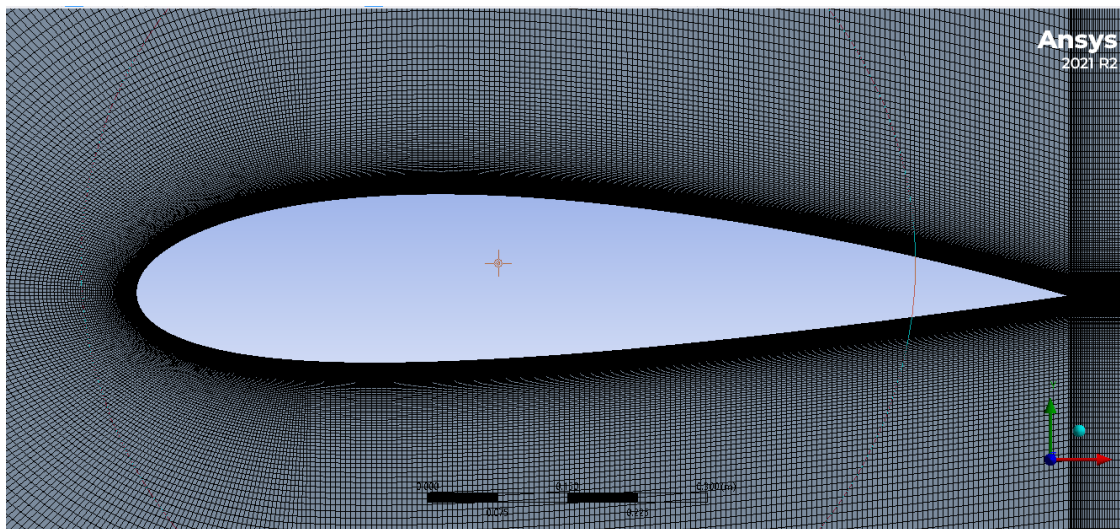


Figure 3: Mesh around the airfoil

Calculations of Y^+

One of the most important subjects in CFD analysis is Mesh independence study. Also known as a mesh convergence study, mesh independency is used to investigate whether simulation results are independent of the underlying mesh or not. It is done by running simulations with different mesh resolutions and checking if the results change.



In this project, according to the previous experiences and also the experts' suggestions, the first layer thickness of boundary layer is calculated; then, with a good precision, it is assumed that mesh independency is achieved by applying this value. An online tool is used to find the first layer thickness, based on the desired y-plus. Since the solver method is set to k-omega in this project, the y-plus must be less than 1. The performance of the airfoil is analyzed in a certain Reynolds number that the reference book determines. By checking the appendix IV of reference [3], the following diagrams are extracted:

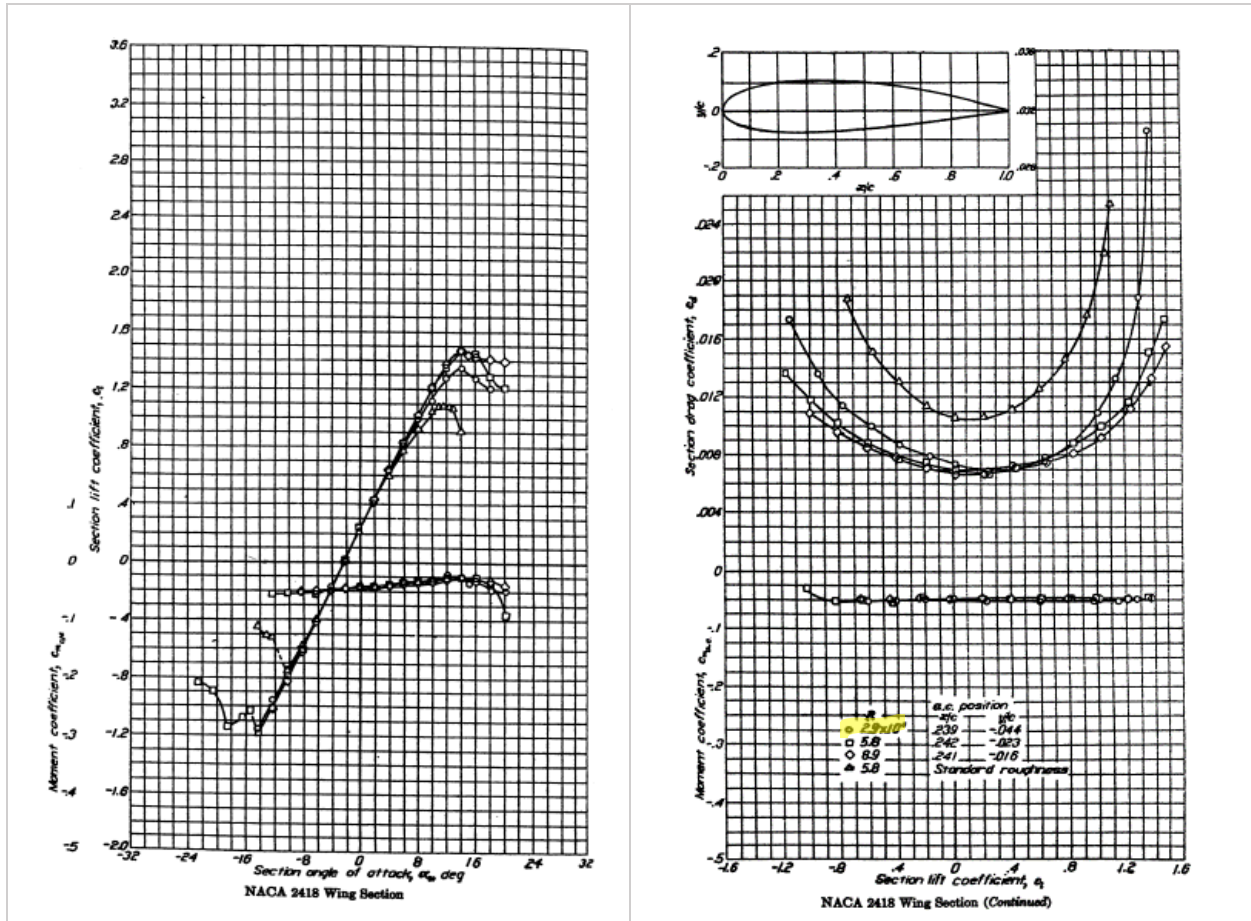


Figure 4: Experimental diagrams for NACA 2418

From these diagrams, the chosen values belong to the first listed Reynolds number, which is shown by circled lines and equals to $Re = 2.9 \times 10^6$. For this Reynolds, the velocity of air can be calculated using the equation:

$$Re = \frac{V.L}{\nu} \rightarrow V = Re \cdot \frac{\nu}{L} = 2.9 \times 10^6 \left(\frac{1.470 \times 10^{-5}}{1} \right) = 42.63 \frac{m}{s}$$



Fluid Mechanics Project



Note that the chord line of the airfoil has 1 meter of length. Also, the properties of air at atmospheric pressure and 15°C is used in this equation, which are extracted from Appendix table. By putting these values in the website, the first layer thickness will be calculated as:

CFD Online
www.cfd-online.com

Home News Forums Wiki Links Jobs

Home > Tools > Y+ Estimation

Y+ Wall Distance Estimation

Input

Freestream velocity: 42.63 [m/s]
 Density: 1.225 [kg/m3]
 Dynamic viscosity: 1.802e-5 [kg/ms]
 Boundary layer length: 1.0 [m]
 Desired Y+ value: 1.0 []

Output

Reynolds number: 2.9e+6 []
 Estimated wall distance: 8.7e-6 [m]

Estimate Wall Distance

Figure 5: First layer thickness

Now it must be ensured that our mesh is smaller than this value. To do so, we zoom in and see:

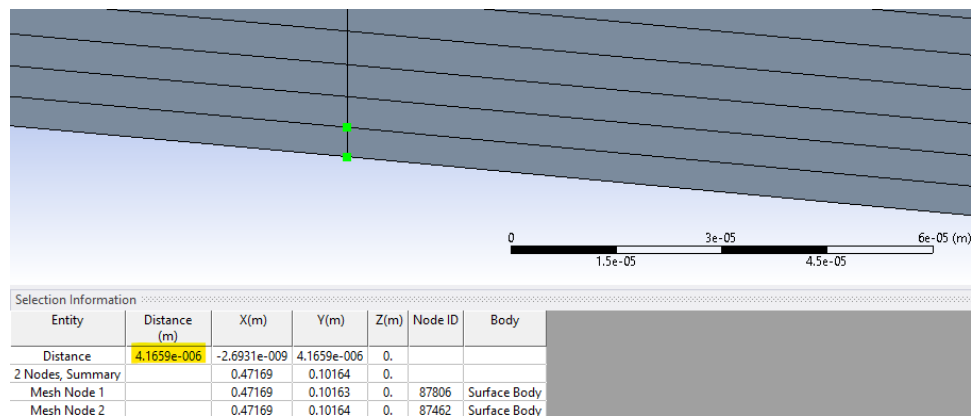


Figure 6: First layer thickness in our mesh



Fluid Mechanics Project



Since $4.17 \times 10^{-6} m$ is smaller than $8.7 \times 10^{-6} m$, our meshing is acceptable. This can also be examined after checking the quality of the mesh using some of its important metrics (see figure 7).

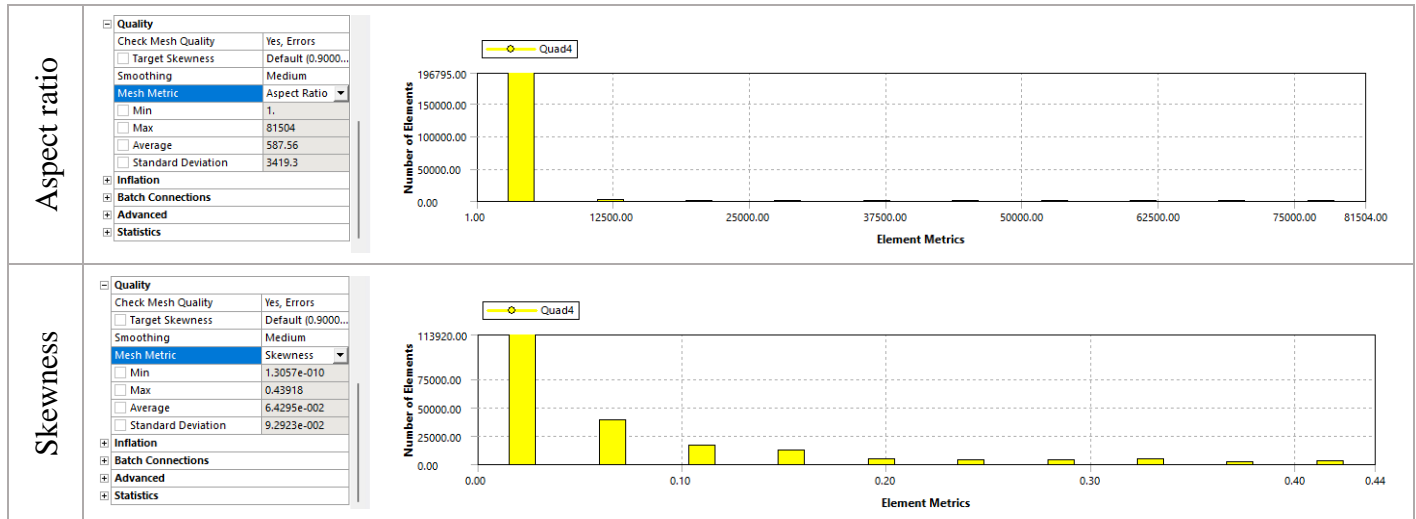


Figure 7: Mesh quality factors

Finally, appropriate names were assigned to every boundary, so that they would be recognizable in setup procedures where specific boundary conditions are allocated to them.



Figure 8: Named selections

Setup

Next step after meshing is setup section which is done in Fluent environment. In this step, we choose to solve the system in planar, pressure based, and steady state. Then we set Viscous solver to *SST k-omega*. After defining the fluid materials (air with properties at 15°C), we specify desired boundary conditions to the named selections we defined in meshing environment. These B.C.s are:



Fluid Mechanics Project



Table 1: Boundary conditions

Surface	Boundary Condition	Setting
Inlet	Velocity inlet	Velocity = 42.63 m/s
Outlet	Pressure outlet	Gauge pressure = 0
Walls	Wall	No slip condition

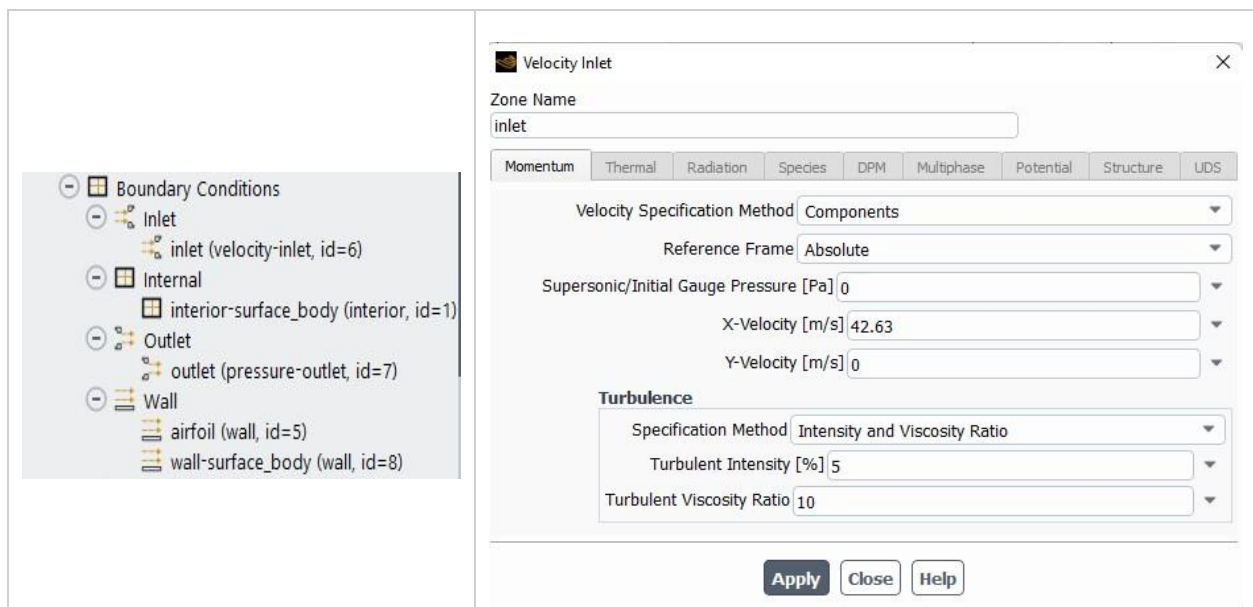


Figure 9: Overview of B.C.s and an example of adjusting inlet B.C.

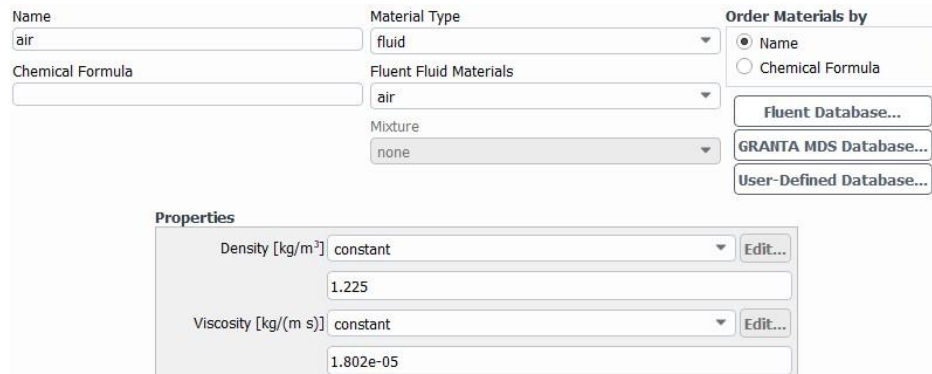


Figure 10: Defining air as the fluid material of system



Fluid Mechanics Project



After boundary conditions, there are a few steps to adjust Solution via *Monitors* sections. The most important command, here, is setting residuals to less than 10^{-6} for all 5 equations. These governing equations of aerodynamical analysis are:

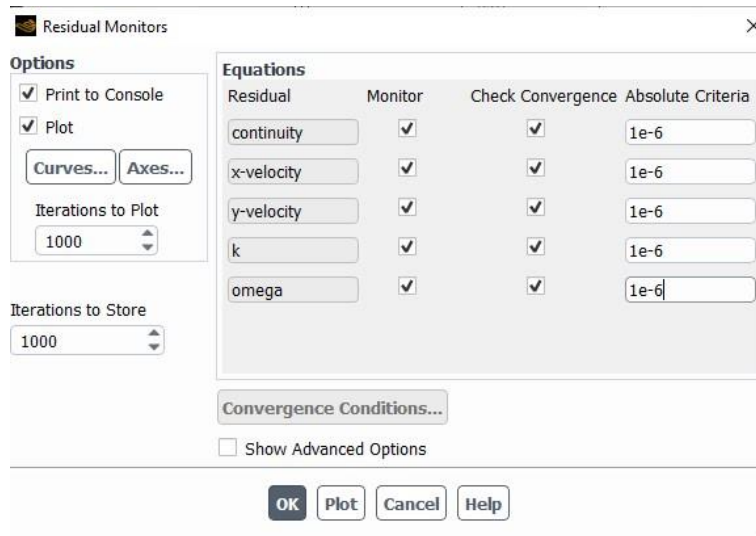


Figure 11: Residuals and equations

We be confident that the system is running in atmospheric pressure by checking the *operating conditions*:

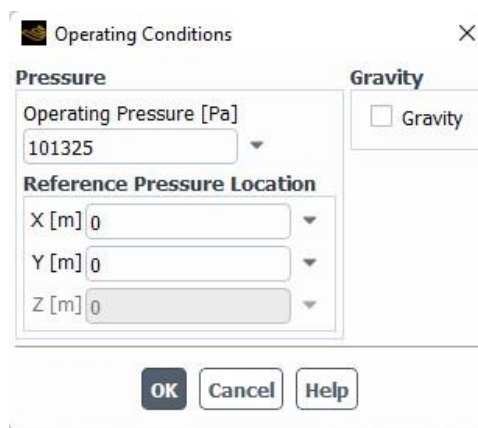


Figure 12: Operating conditions



Fluid Mechanics Project



In order to get the diagrams of drag and lift coefficients of airfoil, we should create new *report's* for them in this way:

The figure shows the ANSYS Fluent interface. The top panel displays the 'Solution' tree with the 'Reports' folder expanded. The 'New' option is selected, leading to a submenu where 'Force Report' is chosen, and then 'Drag...' is selected. The bottom panel shows two side-by-side report definition windows. The left window is for 'Drag Report Definition' and the right is for 'Lift Report Definition'. Both windows have the following settings:

- Name:** drag_coef (for drag) and lift_coef (for lift)
- Options:** Per Zone is unchecked; Average Over (Iterations) is set to 1.
- Report Output Type:** Drag Coefficient (for drag) and Lift Coefficient (for lift).
- Zones:** airfoil and wall-surface_body are selected.
- Force Vector:** X is 1, Y is 0, Z is 1 (for drag); X is 0, Y is 1, Z is 1 (for lift).
- Report Files:** drag_coef-rfile (for drag) and lift_coef-rfile (for lift).
- Report Plots:** drag_coef-rplot (for drag) and lift_coef-rplot (for lift).
- Create:** Report File, Report Plot, and Print to Console are checked; Frequency is set to 1.

Figure 13: Defining reports



Finally, we initialize the setup and then run calculations with 1000 iterations. As a result, it's observed that all equations converge gradually and all of their residuals finally reach less than desired value (10^{-6}); so, it's assured that the obtained results are valid.

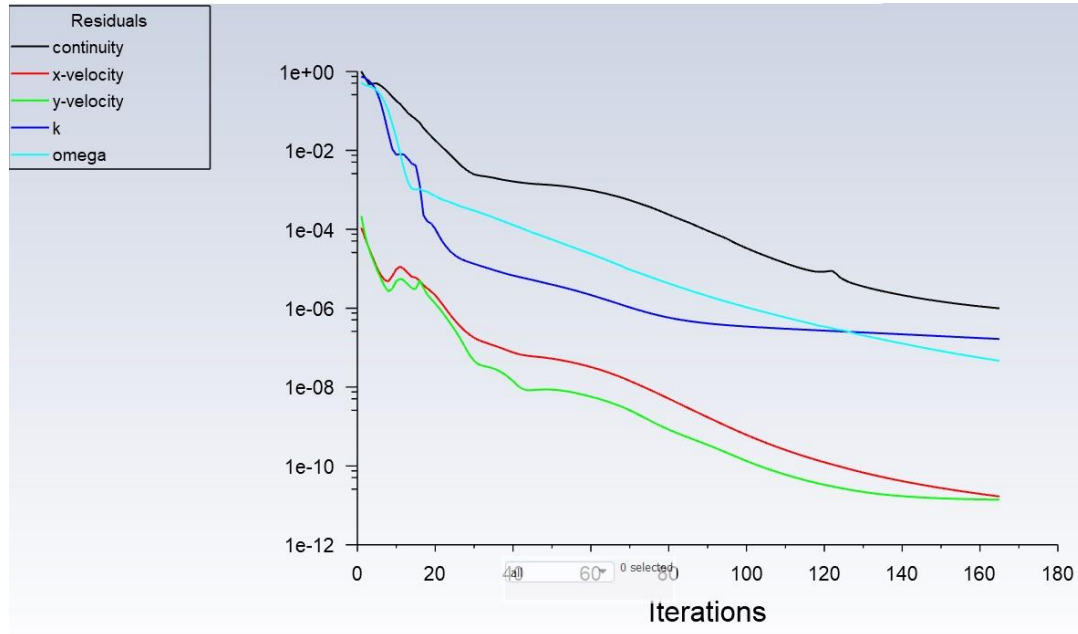


Figure 14: The convergence history for the governing equations

Angle of Attack

All of the mentioned explanations were dedicated to the zero angle of attack. To run the simulation for different angles of attack, the described setup is sufficient only with some small modifications. In this project, to simulate different angles of attack, it's been decided to alter the direction of the flow, instead of rotating the airfoil itself. Thus, for every angle of attack, the velocity input of inlet must be modified with respect to the corresponding angle of attack. For this matter, velocity magnitude must be projected on x and y axes. So, it must be multiplied into the sine and cosine values of different angles of attack, each time.



Fluid Mechanics Project

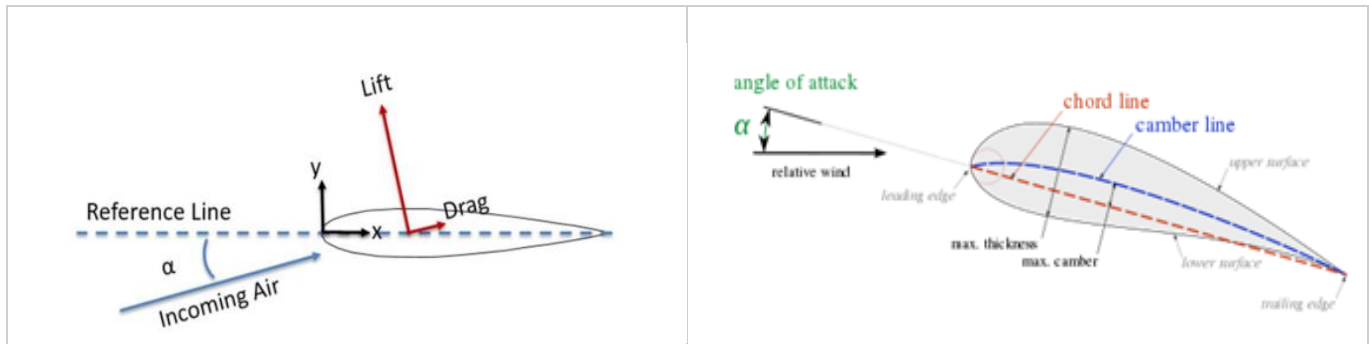


Figure 15: The effect of angle of attack

The effect of angle of attack must also be applied to the *reports*' definition. In fact, the x and y values of *force vector* must be modified by $\sin(\alpha)$ & $\cos(\alpha)$. As an example, for $\alpha = 3^\circ$:

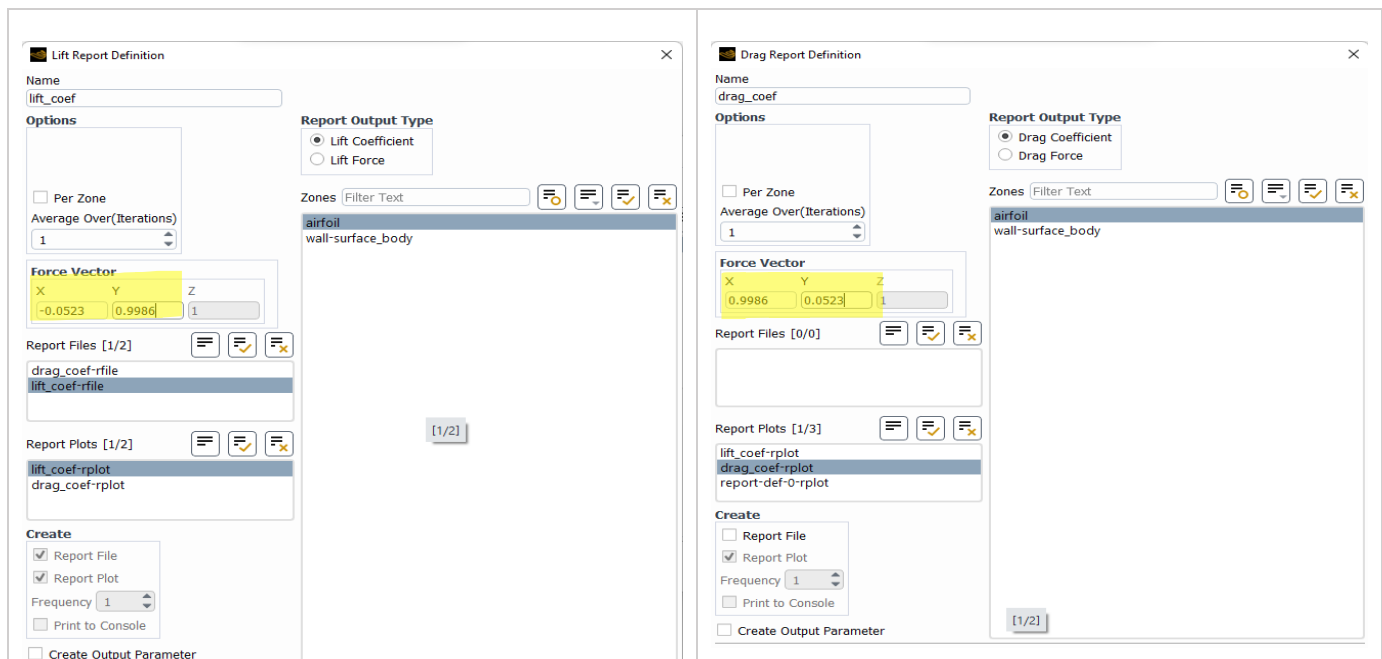


Figure 16: Modification of reports for 3° angle of attack

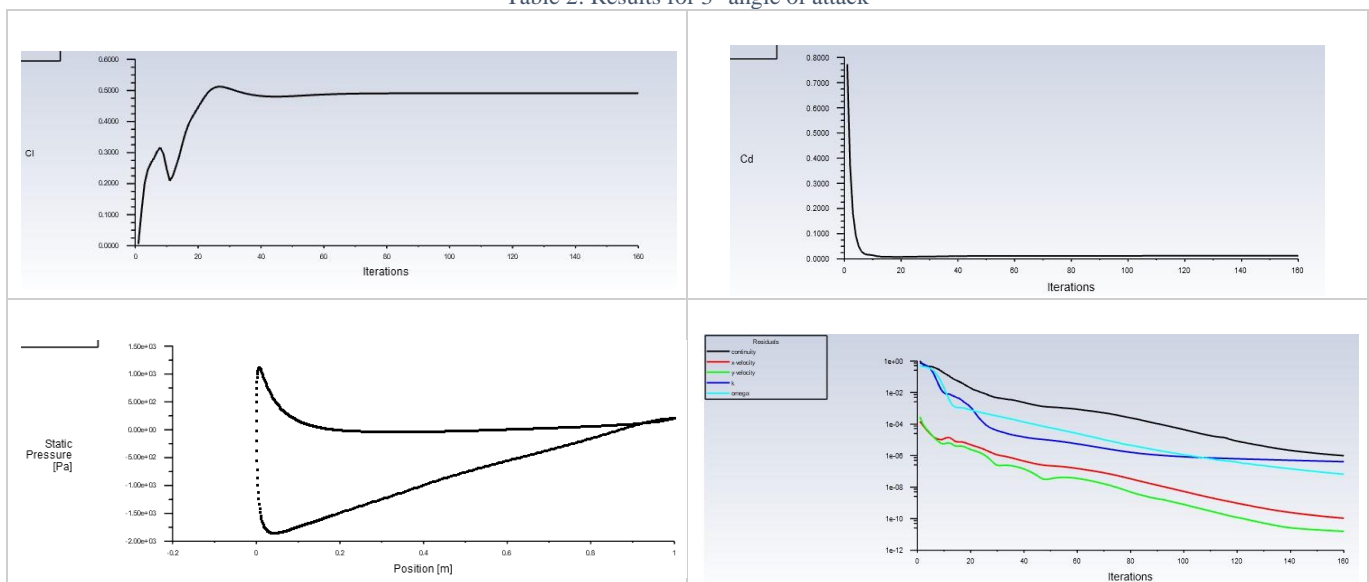


Results and Analysis

After running the simulation for 7 different angles of attack, including $[0^\circ, 3^\circ, 6^\circ, 9^\circ, 12^\circ, 15^\circ$ and $18^\circ]$, all of the required results have been gathered. These results are such as drag coefficient, lift coefficient, and pressure gradient around the airfoil. Here is the summary of all these diagrams:

Individual Plots

Table 2: Results for 3° angle of attack





Fluid Mechanics Project



Table 3: Results for 6° angle of attack

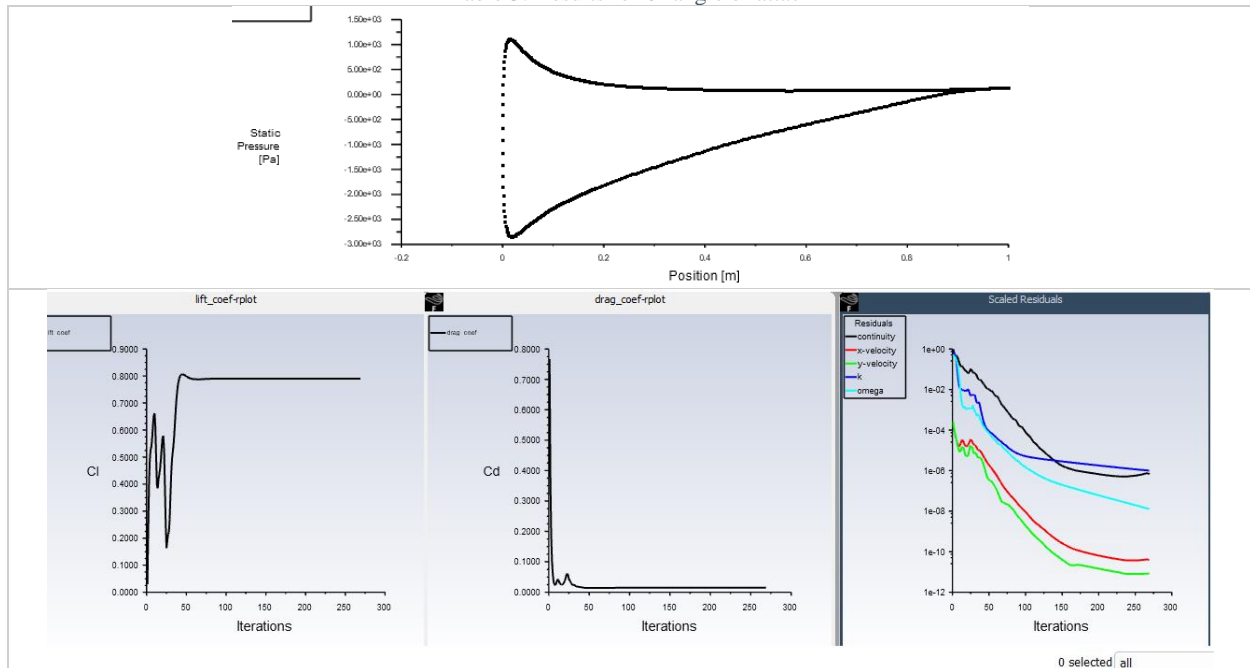
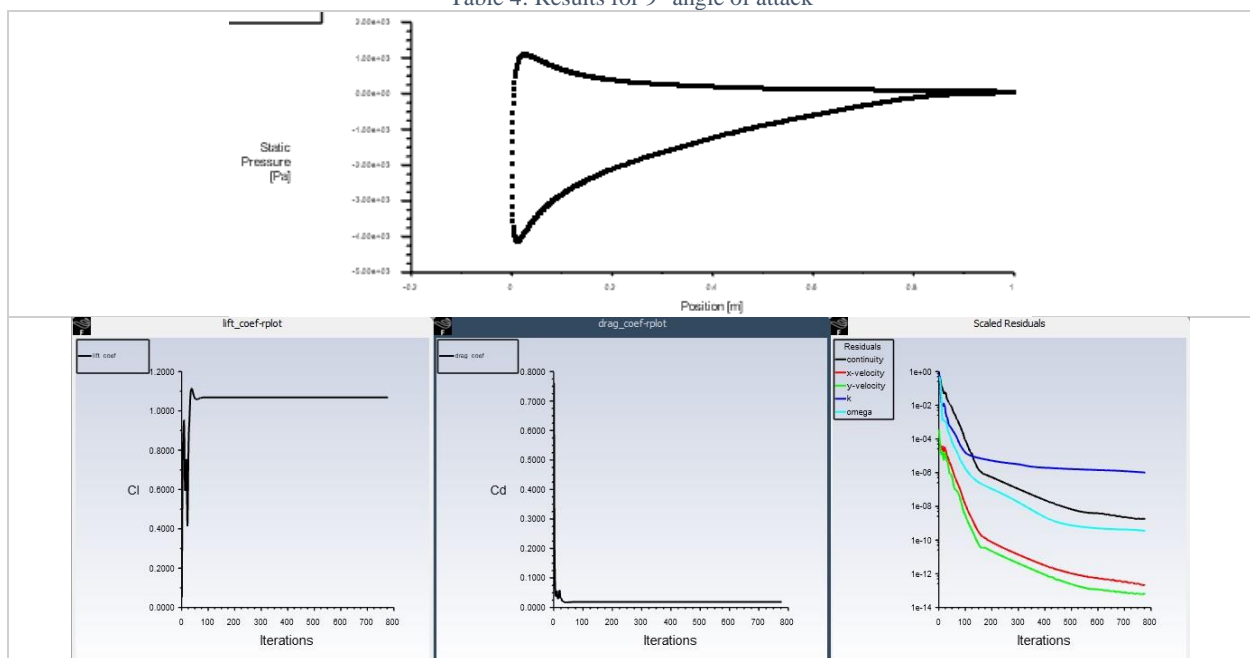


Table 4: Results for 9° angle of attack

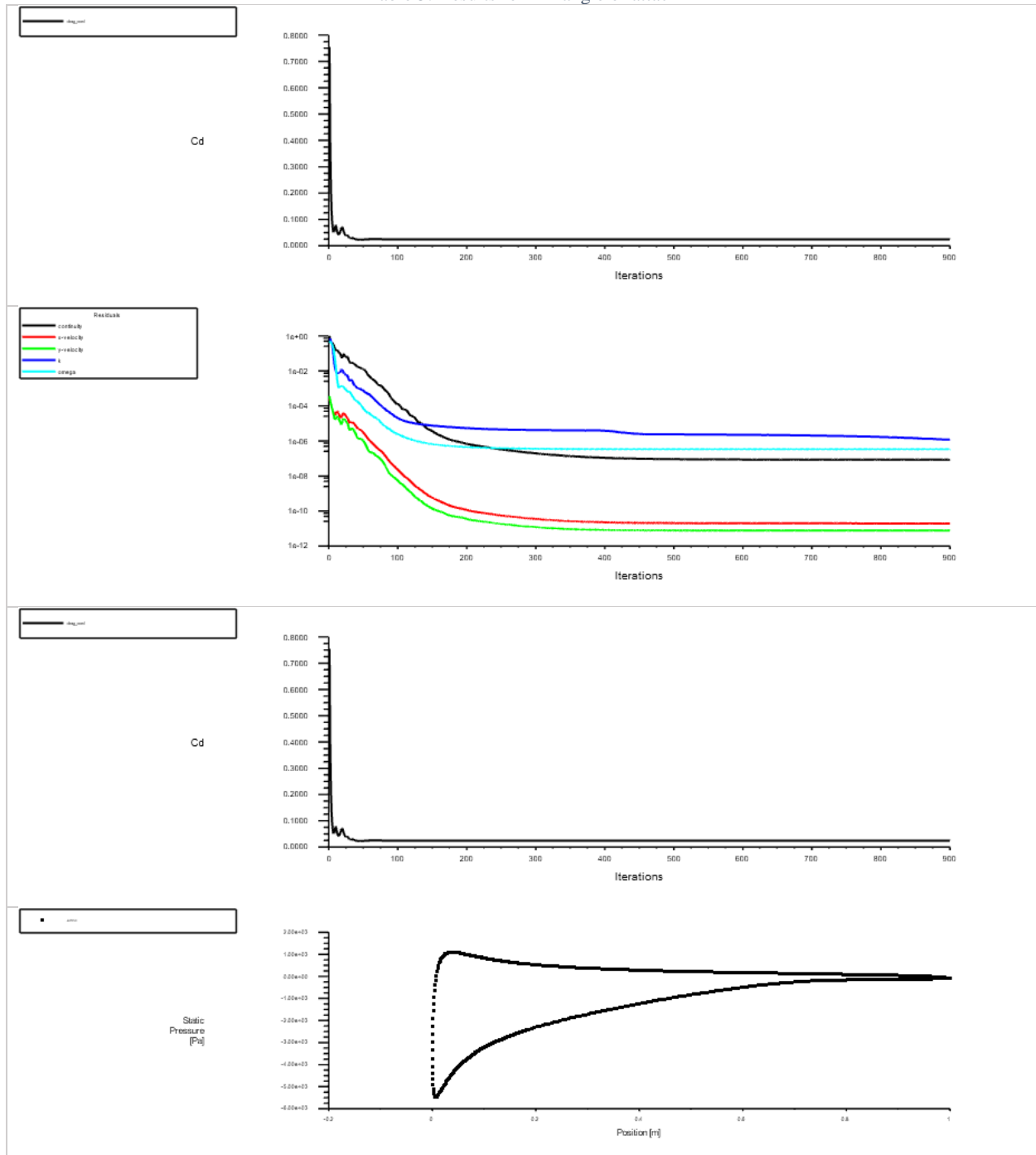




Fluid Mechanics Project



Table 5: Results for 12° angle of attack

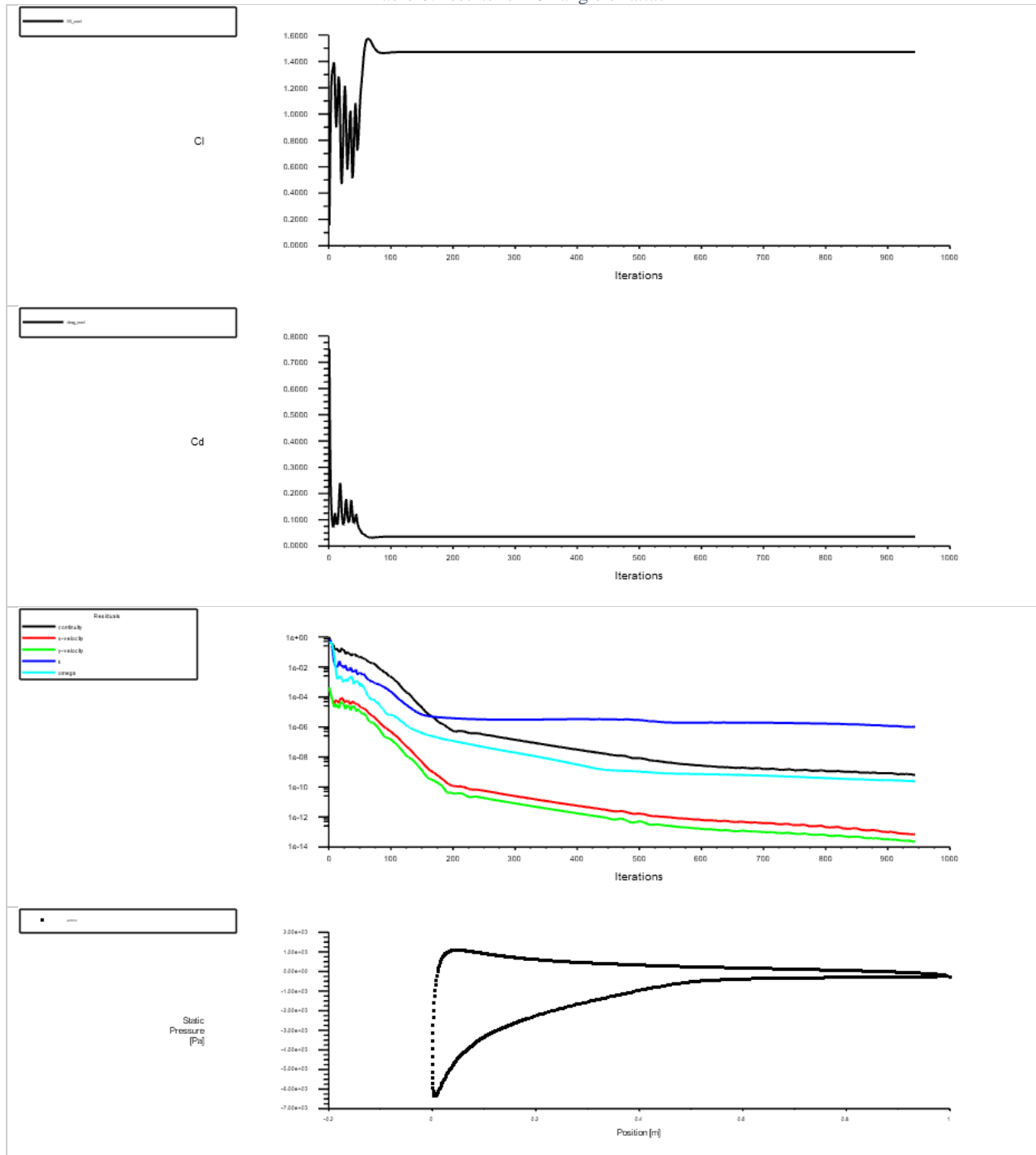




Fluid Mechanics Project



Table 6: Results for 15° angle of attack

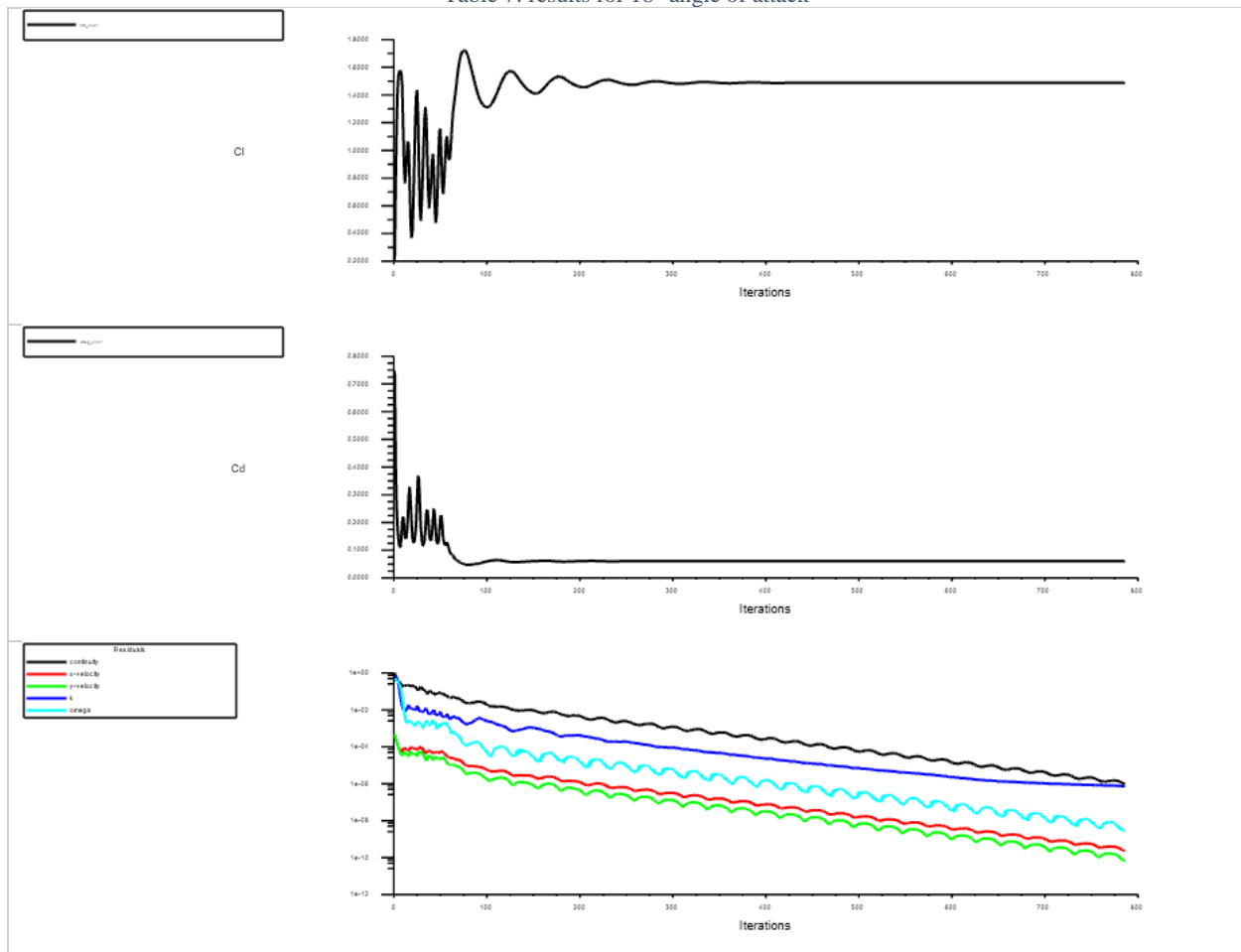




Fluid Mechanics Project



Table 7: results for 18° angle of attack



As it's visible, the fluctuations in all three diagrams for the last angle of attack (18°) are much more considerable in compare to former attack angles. In addition, the lift coefficient of this angle is less than that of previous angles. Thus, it's concluded that the airfoil has reached its stall point at around $\alpha = 15^\circ$. This is also justified by the experimental diagrams.



Overall Plot

Now, by merging all diagrams of pressure gradient, we'll have:

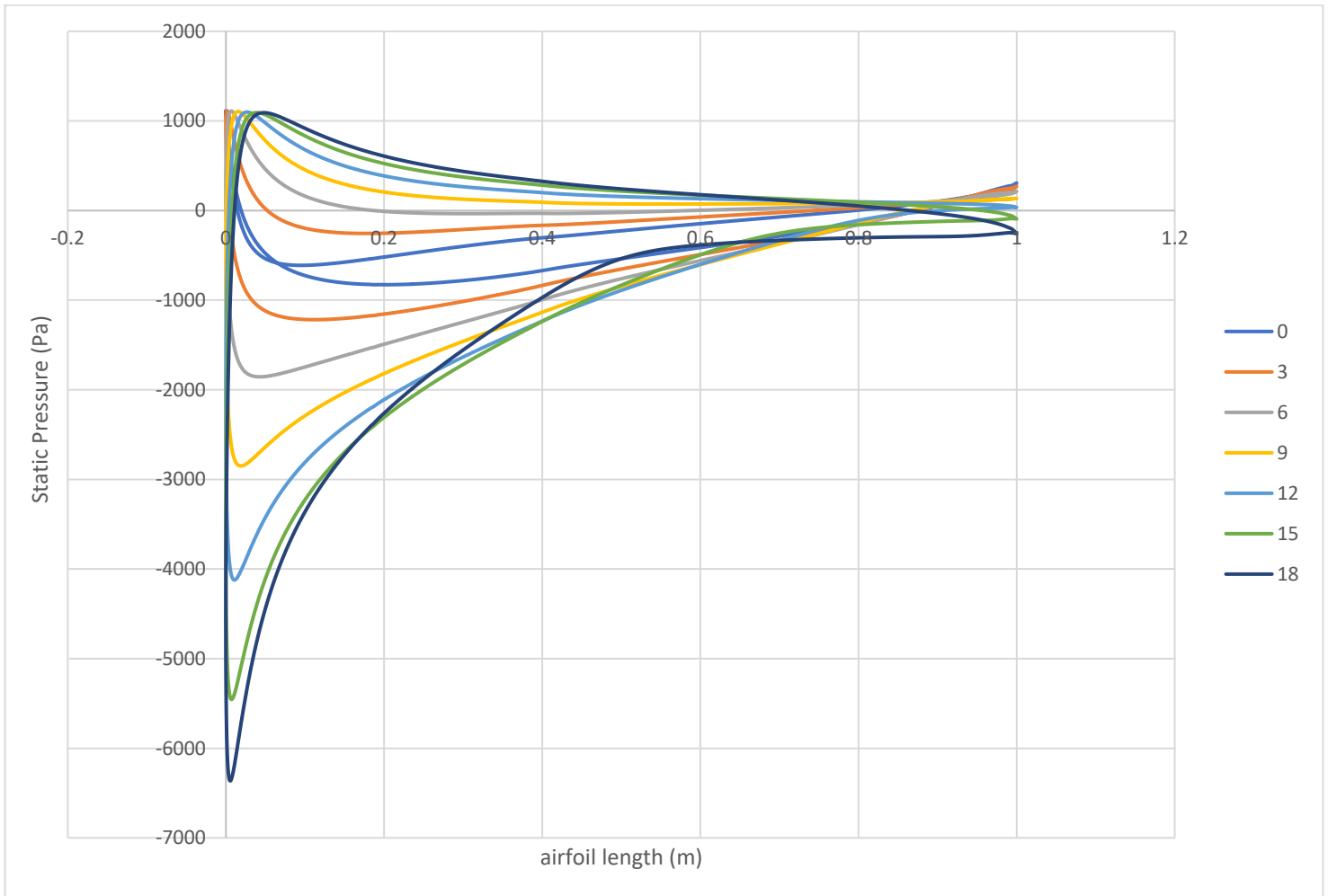


Figure 17: Pressure around airfoil based on its length for 7 different attack angles

It should be noted that above the vertical axis is the bottom of the airfoil and below it is the top plane of airfoil.

Also, as expected, at positive angles of attack, the pressure value below the airfoil is positive and above it is negative.



Lift and Drag Coefficients and Their Errors

While running the simulation for each attack angle, at some point, the calculation stops due to the full convergence of the parameters. This is where the final results for each attack angle should be reported upon. In the following table, these results for drag and lift coefficients have been reported. In addition, the number of iterations until the Fluent had reached these final results has been reported as well.

To validate the results, the experimental values of the mentioned coefficients have been extracted from the reference's tables (see figure 4).

Table 8: Numerical and Experimental Lift and Drag coefficients

Angle of Attack (degrees)	Number of Iterations	Numerical Results		Experimental Results		error (%)	
		Drag Coefficient	Lift Coefficient	C_d	C_l	$\Delta C_d / C_d$	$\Delta C_l / C_l$
0	165	1.0898E-02	1.8571E-01	0.007	0.2	55.7	7.1
3	160	1.2091E-02	4.9121E-01	0.0075	0.55	61.2	10.7
6	269	1.4522E-02	7.9050E-01	0.0085	0.8	70.8	1.2
9	778	1.8398E-02	1.0686E+00	0.012	1.1	53.3	2.9
12	900	2.4688E-02	1.3070E+00	0.019	1.3	29.9	0.5
15	944	3.5934E-02	1.4714E+00	0.036	1.45	0.2	1.5
18	786	6.0638E-02	1.4884E+00	0.015	1.2	304.3	24.0

As it can be observed, there is little error for lift coefficients, while the error of drag coefficients is much more considerable. Another important hint we get from this table is the stall point. As it was explained in the previous section, lift coefficient collapses all of a sudden when the airfoil passes the 15° angle of attack. Therefore, the stall point can be reported as $\alpha = 15^\circ$. Huge error of drag coefficient in the last attack angle can also be justified by the stall phenomenon; since at this point, the flow detaches from the surface, an enormous wake region emerges behind the airfoil. Due to the roughly turbulent characteristics of wake region, to foresee the drag coefficient in this area is kind of impossible.

The 3 most important plots of these results will be shown in the following:



Fluid Mechanics Project

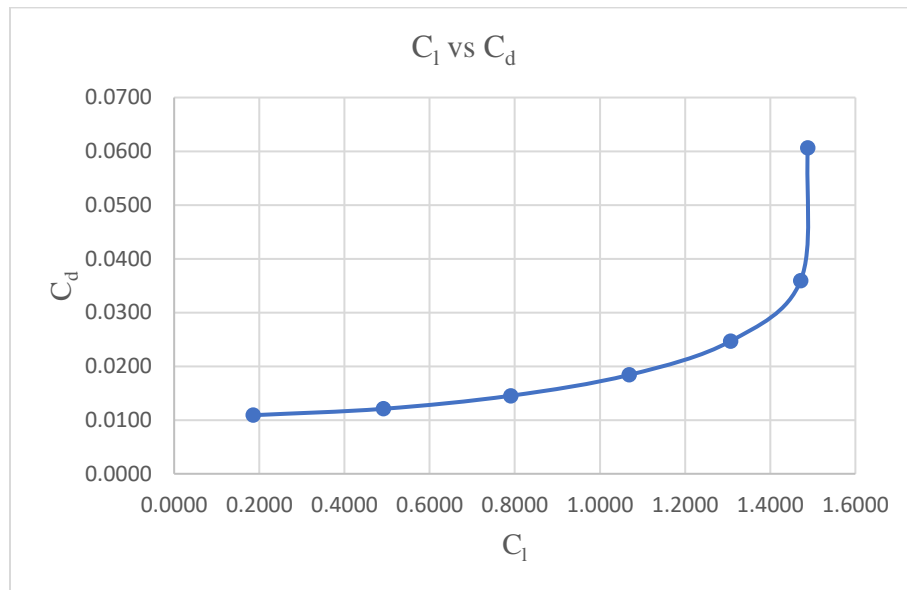


Figure 18: Drag coeff. based on Lift coeff.

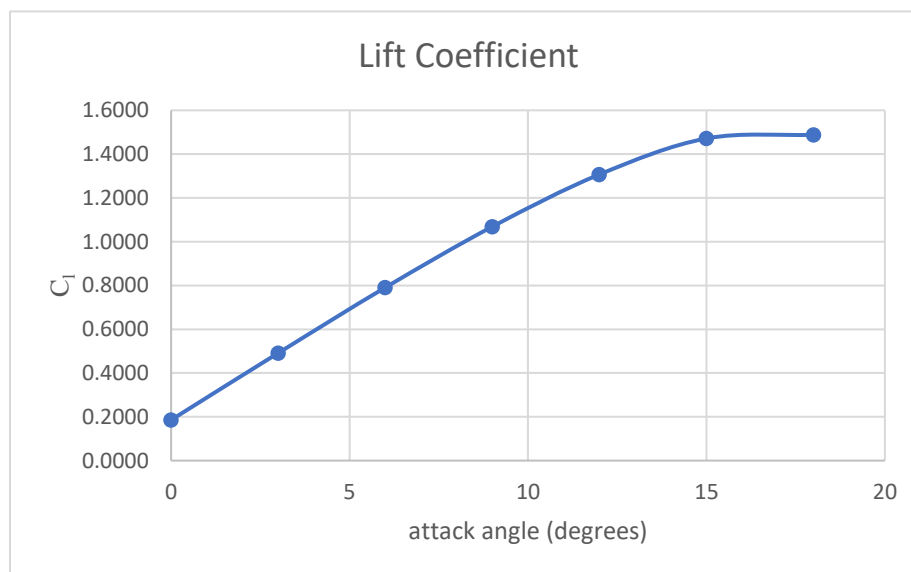


Figure 19: Lift coeff based on attack angle

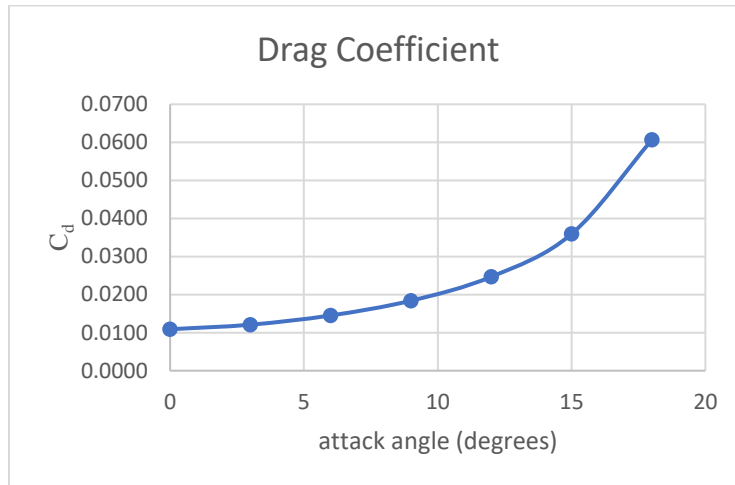


Figure 20: Drag coeff based on attack angle

Researches

Here is a summary of demanded explanations of the project:

a) Prism Layer Meshing and Y^+

Prism layer meshing is a technique used in computational fluid dynamics (CFD) simulations to resolve the boundary layer near the wall. The boundary layer is the thin layer of fluid that forms near the wall of a solid object when a fluid flows over it. The y^+ value is a dimensionless parameter used to determine the size of the first cell near the wall in CFD simulations. The y^+ value is used to ensure that the first cell is small enough to resolve the boundary layer and capture the velocity gradient at the wall.

In computational fluid dynamics (CFD), the boundary layer is the thin layer of fluid that forms near the wall of a solid object when a fluid flows over it. The boundary layer is important because it affects the drag force on the object and can cause separation of the flow.

The y^+ value is a dimensionless parameter used to determine the size of the first cell near the wall in CFD simulations. The y^+ value is used to ensure that the first cell is small enough to resolve the boundary layer and capture the velocity gradient at the wall. The prism layer meshing technique involves adding layers of prismatic cells near the wall to ensure that the first cell is small enough to resolve the boundary layer. The number of prism layers required depends on the Reynolds number and the desired y^+ value.



b) Airfoil Manufacturing Techniques

Airfoils are streamlined bodies that are capable of generating significantly more lift than drag. They are used in wings, sails and propeller blades. There are different methods used to manufacture airfoils such as “airfoil machining” which is used in aviation, aerospace, and energy applications where product quality is an absolute necessity. Another method is the "Spark method" developed by the University of Iceland which involves designing and building two negative/female molds for fabricating the two airfoil halves.

CNC machining is another method used to manufacture airfoils. Hi-Tek Manufacturing uses airfoil machining solutions that power a diverse range of aviation, aerospace, and energy applications where product quality is an absolute necessity. Liechti Engineering's g-Mill is designed for the production of airfoils with 5-axis simultaneous machining technology. Here is a video that shows how to manufacture an airfoil using a CNC machine.

composite layup is another method used to manufacture airfoils. Composite layup is a process of layering materials such as carbon fiber or fiberglass in a mold to create a part. The process involves laying up the composite material in layers and then curing it under heat and pressure. This method is used in the production of one-piece racecar airfoils.

c) Airfoil Simulation Software

ANSYS Fluent is a computational fluid dynamics (CFD) software that is used to simulate fluid flow and heat transfer in complex geometries. It is used to design and optimize airfoils for various applications such as aircraft wings, wind turbines, and hydrofoils. ANSYS Fluent uses numerical methods to solve the governing equations of fluid flow and heat transfer. The software can simulate both steady-state and transient flows.

XFOIL is an interactive program for the design and analysis of subsonic isolated airfoils. It uses a panel method to calculate the pressure distribution on an airfoil. XFOIL can be used to design and analyze airfoils for various applications such as aircraft wings, wind turbines, and hydrofoils.

The main advantage of using ANSYS Fluent over XFOIL is that ANSYS Fluent can simulate fluid flow and heat transfer in complex geometries. This allows for more accurate simulations of real-world conditions. Additionally, ANSYS Fluent can simulate both steady-state and transient flows.



Fluid Mechanics Project



Appendix

Table 9: Properties of air at atmospheric pressure

Properties of air at 1 atm pressure							
Temp. $T, ^\circ\text{C}$	Density $\rho, \text{kg/m}^3$	Specific Heat $c_p, \text{J/kg}\cdot\text{K}$	Thermal Conductivity $k, \text{W/m}\cdot\text{K}$	Thermal Diffusivity $\alpha, \text{m}^2/\text{s}$	Dynamic Viscosity $\mu, \text{kg/m}\cdot\text{s}$	Kinematic Viscosity $\nu, \text{m}^2/\text{s}$	Prandtl Number Pr
-150	2.866	983	0.01171	4.158×10^{-6}	8.636×10^{-6}	3.013×10^{-6}	0.7246
-100	2.038	966	0.01582	8.036×10^{-6}	1.189×10^{-5}	5.837×10^{-6}	0.7263
-50	1.582	999	0.01979	1.252×10^{-5}	1.474×10^{-5}	9.319×10^{-6}	0.7440
-40	1.514	1002	0.02057	1.356×10^{-5}	1.527×10^{-5}	1.008×10^{-5}	0.7436
-30	1.451	1004	0.02134	1.465×10^{-5}	1.579×10^{-5}	1.087×10^{-5}	0.7425
-20	1.394	1005	0.02211	1.578×10^{-5}	1.630×10^{-5}	1.169×10^{-5}	0.7408
-10	1.341	1006	0.02288	1.696×10^{-5}	1.680×10^{-5}	1.252×10^{-5}	0.7387
0	1.292	1006	0.02364	1.818×10^{-5}	1.729×10^{-5}	1.338×10^{-5}	0.7362
5	1.269	1006	0.02401	1.880×10^{-5}	1.754×10^{-5}	1.382×10^{-5}	0.7350
10	1.246	1006	0.02439	1.944×10^{-5}	1.778×10^{-5}	1.426×10^{-5}	0.7336
15	1.225	1007	0.02476	2.009×10^{-5}	1.802×10^{-5}	1.470×10^{-5}	0.7323
20	1.204	1007	0.02514	2.074×10^{-5}	1.825×10^{-5}	1.516×10^{-5}	0.7309
25	1.184	1007	0.02551	2.141×10^{-5}	1.849×10^{-5}	1.562×10^{-5}	0.7296
30	1.164	1007	0.02588	2.208×10^{-5}	1.872×10^{-5}	1.608×10^{-5}	0.7282



References

- [1] Fox, R. W., Pritchard, P. J., and McDonald, A. T., Introduction to Fluid Mechanics, 8th edition, Wiley, 2011.
- [2] Cengel, Yunus A.; Ghajar, Afshin J.; Heat and Mass Transfer: FUNDAMENTALS AND APPLICATIONS; SIXTH EDITION; McGraw-Hill Education; 2020.
- [3] I. H. Abbott and A. E. Von Doenhoff, Theory of wing sections, including a summary of airfoil data. Courier Corporation, 1959.
- [4] Anthony T; ANSYS Fluent NACA 4412 (or NACA 0012) 2D airfoil CFD Tutorial with Experimental Validation (2021); <https://www.youtube.com/watch?v=nzvEvLCxOss>.
- [5] Cillian Thomas; NACA2412 Tutorial in ANSYS Fluent (Student Version) - Lift, Drag, Angle of Attack; <https://www.youtube.com/watch?v=3i9Ryq-m1HA>.
- [6] Klaus A. Hoffmann (Author), Steve T. Chiang, Computational Fluid Dynamics for engineers, 4th edition, Engineering Education System, 2000.
- [7] Airfoil - Wikipedia. <https://en.wikipedia.org/wiki/Airfoil>.
- [8] Airfoil Design 101: What Is an Airfoil? - National Aviation Academy. <https://www.naa.edu/airfoil-design/>.
- [9] Airfoil - Aircraft Aerodynamics. <https://www.aircraftsystemstech.com/p/airfoil.html>.
- [10] Airfoil Tools. <http://airfoiltools.com/>.
- [11] Airfoil database search. <http://airfoiltools.com/search/index>.
- [12] Airfoils and Their Applications - Aircraft Basic Science; <https://www.oreilly.com/library/view/aircraft-basic-science/9780071799171/ch4.html>.
- [13] Airfoil and Turbine Blade Machining - Hi-Tek Manufacturing. <https://www.hitekmg.com/solutions/airfoil-and-turbine-blade-machining>.
- [14] University of Iceland develops process for manufacturing one-piece; <https://www.compositesworld.com/articles/university-of-iceland-develops-process-for-manufacturing-one-piece-racecar-airfoils>.
- [15] "airfoil" 3D Models to Print - yeggi. <https://www.yeggi.com/q/airfoil/>.
- [16] How To: 3D Print an Airfoil - YouTube. <https://www.youtube.com/watch?v=FEpKYidS8xw>.
- [17] A perfect fit! - Aerospace Manufacturing. <https://www.aero-mag.com/a-perfect-fit>.
- [18] How to manufacture an aerofoil using a CNC machine - YouTube. <https://www.youtube.com/watch?v=GbQHZHiZWwA>.