

# NACA 0012 AIRFOIL TURBULENCE VALIDATION USING ANSYS FLUENT

Dinesh Poluru Baskar

HOCHSCHULE FURTWANGEN UNIVERSITY Germany. [dineshpb0812@gmail.com](mailto:dineshpb0812@gmail.com)

## Table of Contents

Abstract.....	2
Introduction .....	2
NACA 0012 .....	2
Spalart – Allmaras Turbulence Model.....	3
Procedure.....	4
Creating the coordinates for NACA 0012:.....	4
Creating the Geometry: .....	5
Meshing: .....	5
Setup: .....	6
Results and Discussion .....	7
Angle of attack 0 degree: .....	7
Angle of attack 10 degree: .....	9
Conclusion.....	11

## Figure of Contents

Figure 1: NACA 0012 airfoil [ (Airfoil Tools, n.d.)]. .....	2
Figure 2: The Spalart – Allmaras model. ....	3
Figure 3: Mesh around the wall region. ....	4
Figure 4: Transport equation [ (Wikipedia, n.d.)]. ....	4
Figure 5: NACA 0012 airfoil. ....	5
Figure 6: Mesh of NACA 0012. ....	5
Figure 7: y plus illustration. ....	6
Figure 8: Contour. ....	6
Figure 9: Pressure contour for 0-degree $\alpha$ . ....	7
Figure 10: Velocity contour for 0-degree $\alpha$ . ....	7
Figure 11: Drag co-efficient. ....	8
Figure 12: Lift co-efficient. ....	8
Figure 13: $C_d$ computed.....	8
Figure 14: $C_l$ computed. ....	9
Figure 15: NASA data validation[ (Rumsey, 2020)]......	9
Figure 16: Pressure contour for 10-degree $\alpha$ . ....	9
Figure 17: Velocity contour for 10-degree $\alpha$ . ....	10
Figure 18: Drag co-efficient. ....	10
Figure 19: Lift co-efficient. ....	10
Figure 20: $C_d$ computed.....	11
Figure 21: $C_l$ computed. ....	11

## Abstract

The following report is a complete explanation of turbulence modelling for NACA 0012 airfoil. Ansys Fluent software is used for this purpose and all the related graphs and images are explained clearly in this report. This report is not based on any new project rather it gives detailed insight comparison for the analysis between the NASA validation and this validation data.

## Introduction

This project is based on validation of pressure and velocity profiles at the airfoil using ANSYS Fluent software. NACA 0012 airfoil is used for this purpose because of its symmetric shape and the line of chord is symmetric to the airfoil, which is better in case of analyzing in a small computer. Airfoil coordinates are taken from the NACA website and analysis is done by using **Spalart – Allmaras** turbulence model.

## NACA 0012

NACA 0012 is commonly used by many industries as the test case profile for many numerical fluid mechanics. This airfoil is symmetric to the position and has no camber angle related to it. The first to digits of the number denotes the camber angle of the structure which in this case is zero, while the other two numbers denote the ratio of profile thickness to the length of the profile chord.

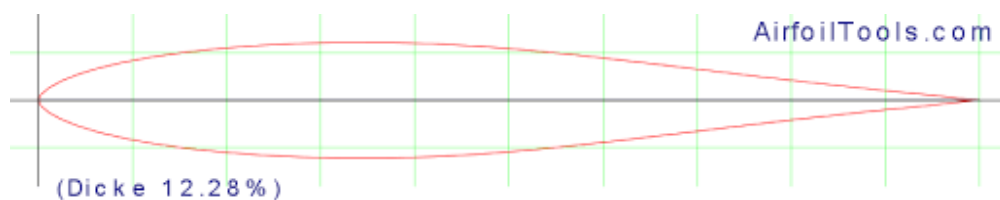


Figure 1: NACA 0012 airfoil [ (Airfoil Tools, n.d.)].

## Spalart – Allmaras Turbulence Model

The Spalart – Allmaras model is specifically designed for aeronautics purposes. This model is better than  $k - \omega$  and  $k - \omega$  SST because they fail to work better at the boundary layer condition and at adverse pressure gradients. A paper published by G. Kalitzin, G. Medic, G. Iaccarino and P. Durbin on the “Near-wall behavior of RANS turbulence models and implication for wall function” proved that the turbulence at the near wall condition varies  $y^+$  to the power 4 and then above the boundary layer the curve gets linear.

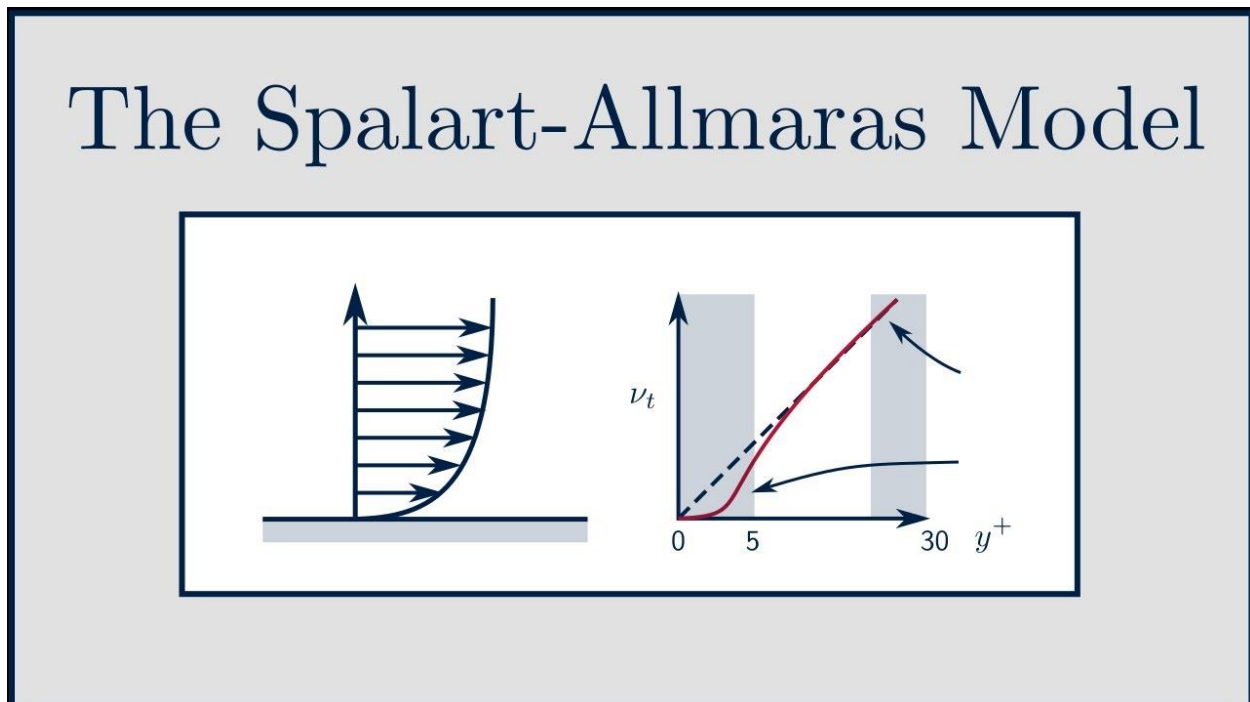


Figure 2: The Spalart – Allmaras model.

As we can see from the above logarithmic graph, near the wall region the turbulence is massive and eventually it becomes linear through out the flow. For this reason, while using CFD analysis a lot of cells should be present near the wall region as suppose to the outer region.

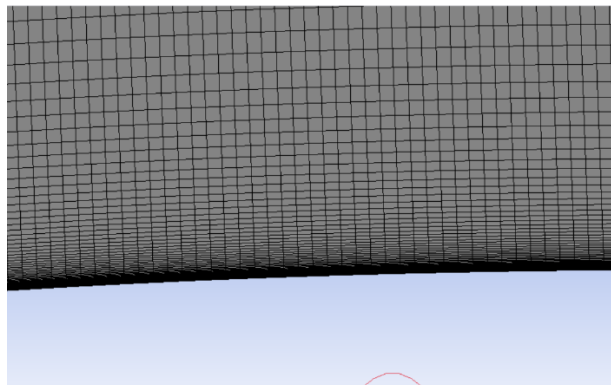


Figure 3: Mesh around the wall region.

As we can see from the graph near the wall region the line actually curves (non-linear behavior) and at the end it tries to gain back its linearity. But with the use of Spalart – Allmaras model when we solve for ***nu tilde***, we can get close to the dotted-lines. This can be achieved by solving the transport equation for ***nu tilde***. The equation for the Spalart – Allmaras is given as:

$$\frac{\partial \tilde{\nu}}{\partial t} + u_j \frac{\partial \tilde{\nu}}{\partial x_j} = C_{b1} [1 - f_{t2}] \tilde{S} \tilde{\nu} + \frac{1}{\sigma} \{ \nabla \cdot [(\nu + \tilde{\nu}) \nabla \tilde{\nu}] + C_{b2} |\nabla \tilde{\nu}|^2 \} - \left[ C_{w1} f_w - \frac{C_{b1}}{\kappa^2} f_{t2} \right] \left( \frac{\tilde{\nu}}{d} \right)^2 + f_{t1} \Delta U^2$$

Figure 4: Transport equation [ (Wikipedia, n.d.)].

This new variable ***nu tilde*** is easier to solve numerically than ***nu t***, as the solution is likely to be close to linear. The first term on the right-hand side of the equation is responsible for the generation of the turbulence in the flow field. This generation is done in the place where high shear is occurring (near the wall region). Near the wall region the vector of one fluid particle moves faster than the other which in turn creates this turbulence in the flow field. The second term is the diffusion term which is responsible for the mixing of the fluids from different sources. Every transport equation contains a diffusion term and a convective term. There is a non-linear term at the end of the diffusion term which is not a usual term on any transport equation. The last term is the destruction term which is responsible for destructing the presence of turbulence at the wall region by damping the viscosity at the near wall region. This is indicated by a negative symbol because it's going to reduce the turbulence (***nu tilde***) near the wall. The “d” represents the distance from the wall, as we go near the wall the damping is increased as a result the turbulence is reduced.

## Procedure

The simulation procedure for the NACA 0012 is a straightforward process in which every step is explained clearly down below. The boundary conditions which are being used are relatively compared from the NASA website where they used the same result for the analysis.

### Boundary Conditions:

- NACA 0012 airfoil
- Pointed trailing edge.
- Incompressible flow ( $Ma < 0.3$ ).
- Reynolds number 6 million
- $Y^+$  less than 1.0

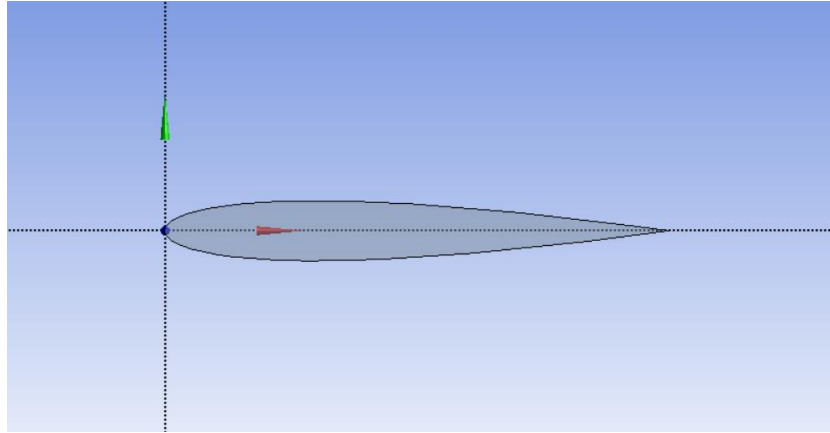
### Creating the coordinates for NACA 0012:

The NACA 0012 airfoil can be created by using an online airfoil builder where the specific type of airfoil can be created with the help of coordinates point. The link is given below. The coordinates are then stored in an Excel file where it can be saved as a text documented file and can be transferred to the Ansys for the simulation process.

<http://airfoiltools.com/airfoil/naca4digit>

### Creating the Geometry:

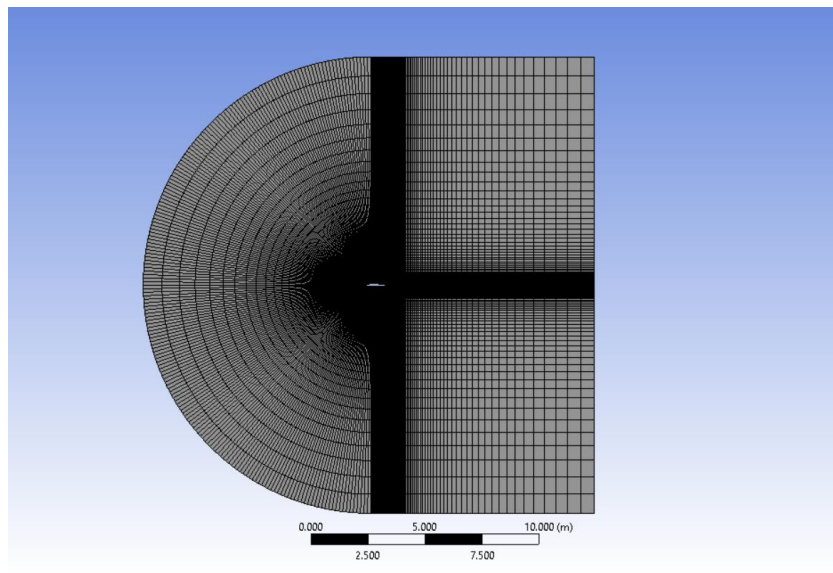
The file is imported to the Ansys geometry and the outer region is created with 20 chords away from the airfoil. NASA did this in 500 chords which can be run easily by NASA scientist in their powerful computers. But for simple home computers 20 chords is fine enough to get the results as accurate as possible.



*Figure 5: NACA 0012 airfoil.*

### Meshing:

In this method, the outer field is divided by 6 parts and the inside part is provided with edge sizing from the boundary of the airfoil finer to the looser type. And then the outer geometry is selected and face meshing is provided.



*Figure 6: Mesh of NACA 0012.*

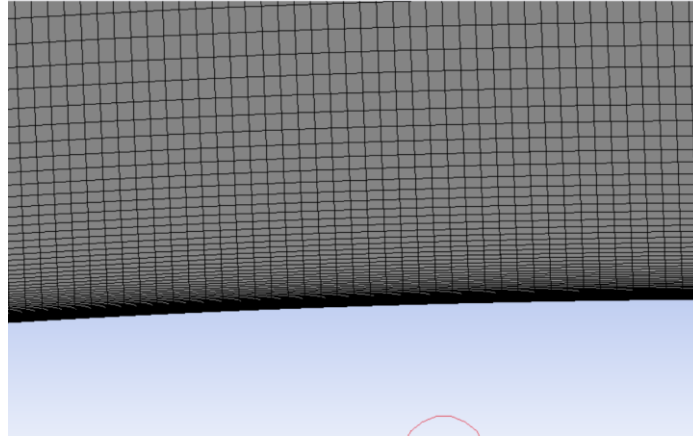


Figure 7: *y plus* illustration.

As we can see from the above picture the *y plus* values is finer near the boundary and lose as the geometry goes up.

### Setup:

The setup case for NACA airfoil consists of two types of validation. The angle of attack has been changed and compared the values with the NASA validation sheet. In the fluent solver the required boundary conditions has been given by choosing the spalart – Allmaras turbulence model with the velocity of 88.65 m/s and with Mach number 0.3.

The drag and lift gradients have been observed and compared it with the original NASA validation. The graphs for drag and lift has been computed as the solution is running for both the angle of attacks.

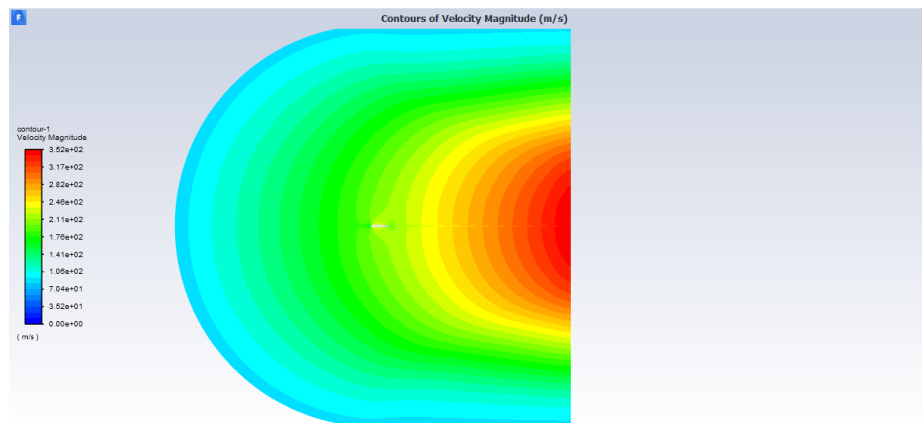


Figure 8: *Contour*.

## Results and Discussion

The simulation is run for 0 and 10 degree of angle of attack where the initial velocity and all the parameters are remained constant. The results produced for both the angle of attack has been observed and displayed below with the comparison of NASA validation data.

### Angle of attack 0 degree:

At this point the x and y component of the airfoil has not changed and run the simulation in the default setting. Only the velocity magnitude has been given in the inlet condition tab.

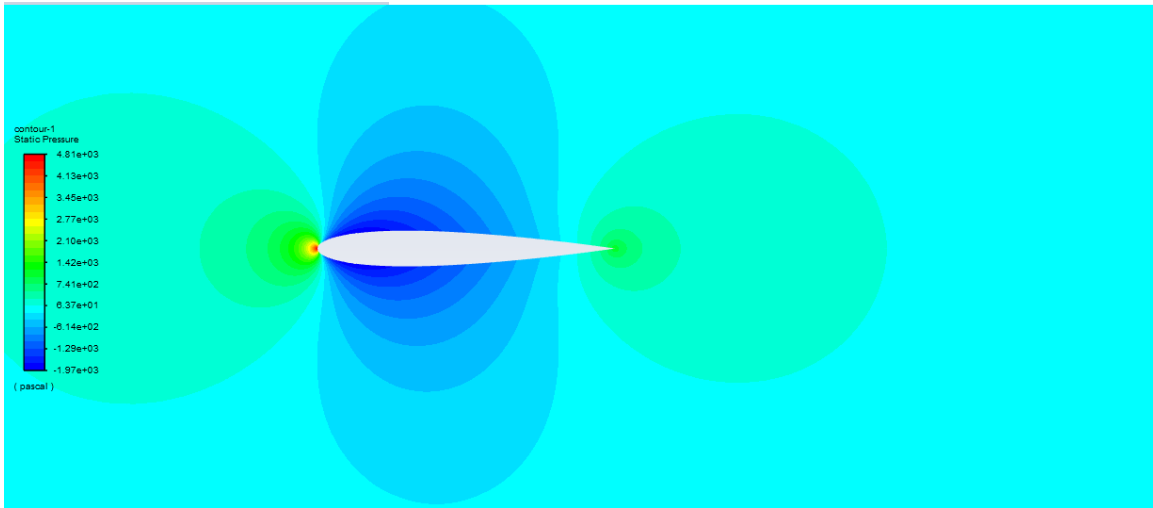


Figure 9: Pressure contour for 0-degree  $\alpha$ .

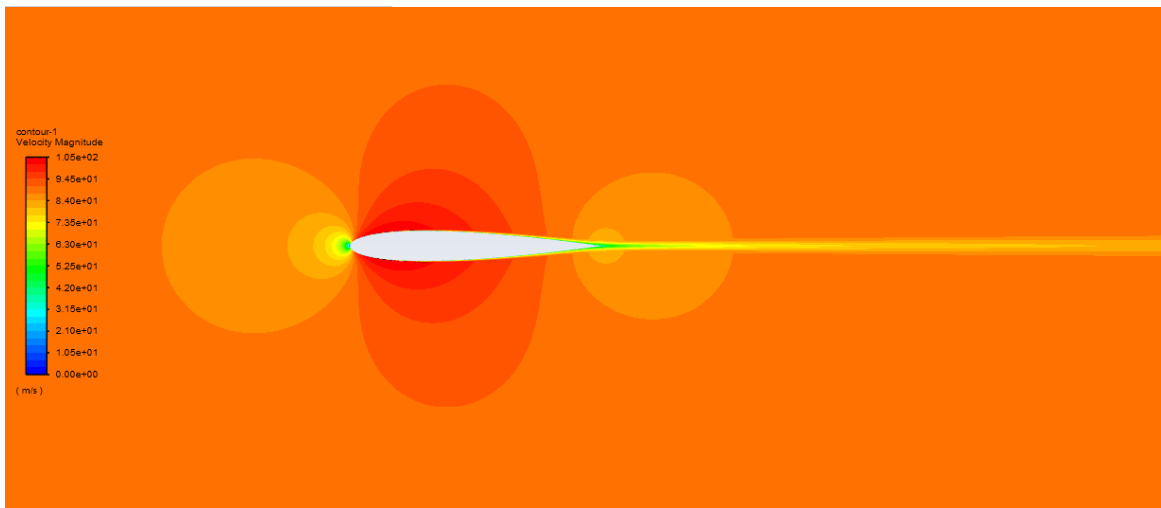


Figure 10: Velocity contour for 0-degree  $\alpha$ .



Now that we have seen the pressure and velocity contour figures, it clearly shows that at the point of attack the pressure is more as the air molecules moves towards the wing side the velocity increases as a result the pressure decreases. Now we will look at the drag and lift components of the model.

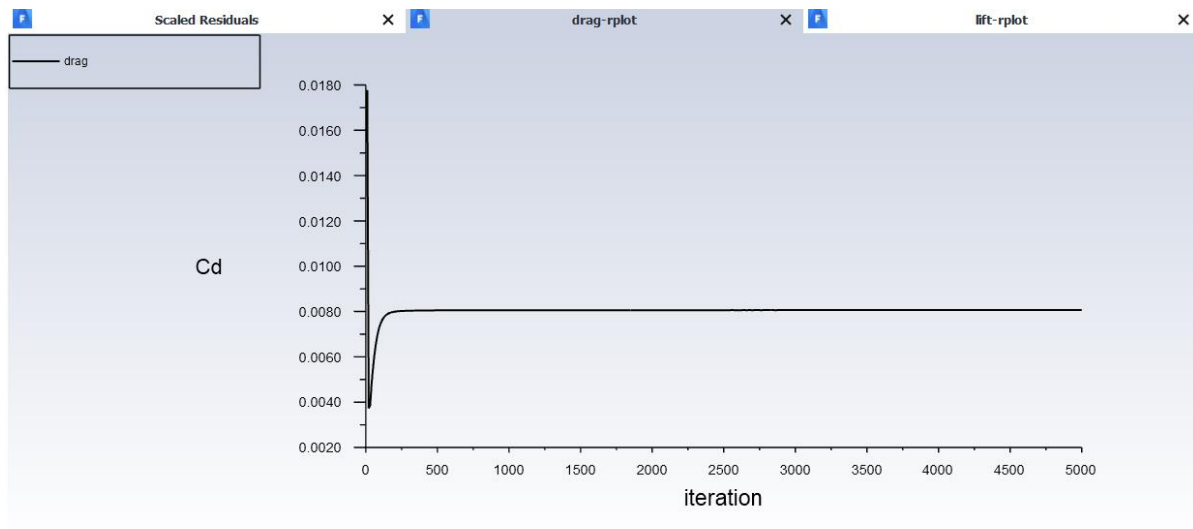


Figure 11: Drag co-efficient.

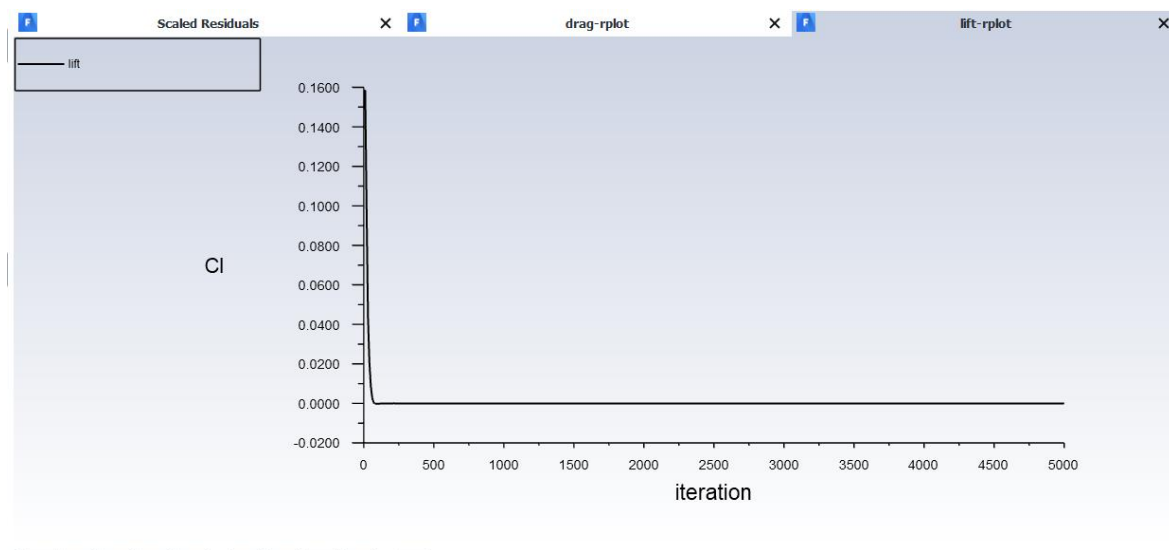


Figure 12: Lift co-efficient.

Lift and drag co-efficient are almost straight line as at the point of attack there is little lift and drag and around the chord it becomes constant. The computed values for the drag and lift are shown below.

Cd	()
airfoil	0.0080618309

Figure 13:  $C_d$  computed.

CL	()
airfoil	-3.7786094e-06

Figure 14:  $C_l$  computed.

CODE	CL (alpha=0)	CL (alpha=10)	CL (alpha=15)	CD (alpha=0)	CD (alpha=10)	CD (alpha=15)
CFL3D	approx 0	1.0909	1.5461	0.00819	0.01231	0.02124
FUN3D	approx 0	1.0983	1.5547	0.00812	0.01242	0.02159
NTS	approx 0	1.0891	1.5461	0.00813	0.01243	0.02105
JOE	approx 0	1.0918	1.5490	0.00812	0.01245	0.02148
SUMB	approx 0	1.0904	1.5446	0.00813	0.01233	0.02141
URNS	approx 0	1.1000	1.5642	0.00830	0.01230	0.02140
GGNS	approx 0	1.0941	1.5576	0.00817	0.01225	0.02073

Figure 15: NASA data validation[ (Rumsey, 2020)].

From the above given pictures, we can see that the computed value for  $C_d$  and  $C_l$  is 0.008061 and -3.7786e-06 respectively is really close to the NASA validation data as 0.00819 and approx. 0 which is approximately close to the NASA data.

### Angle of attack 10 degree:

In this method the x component and y component are changed for 10 degree of angle of attack. The angle of  $\sin 10$  and  $\cos 10$  has been calculated and provided on the x and y component on the setup side and the results are observed.



Figure 16: Pressure contour for 10-degree  $\alpha$ .

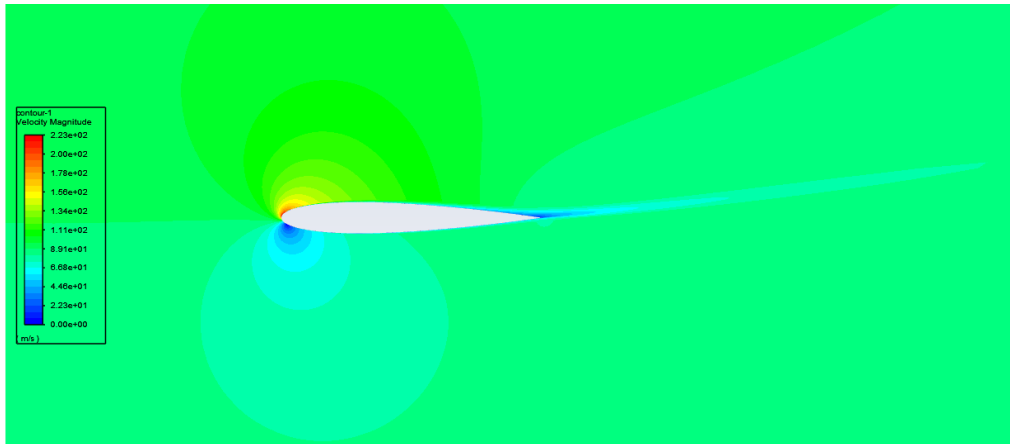


Figure 17: Velocity contour for 10-degree  $\alpha$ .

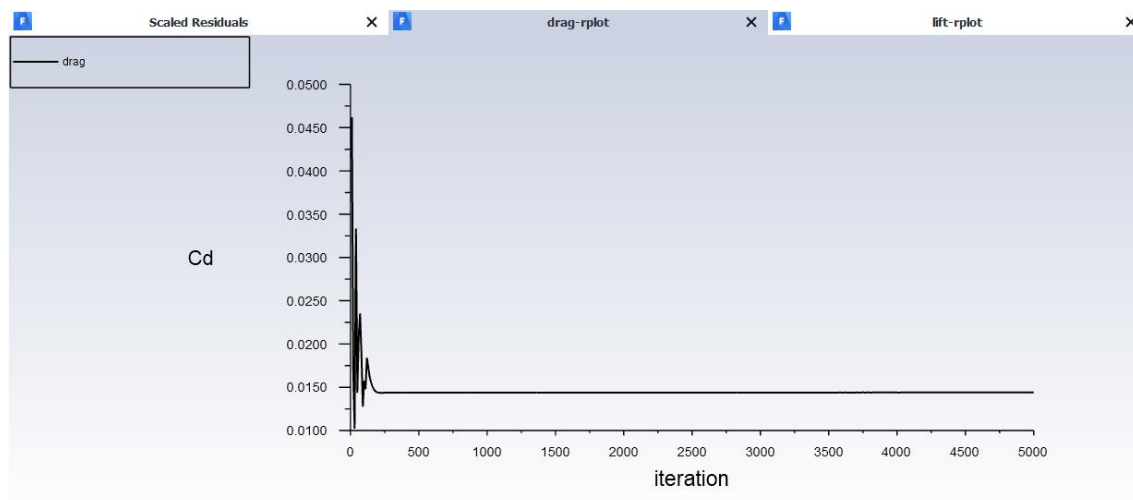


Figure 18: Drag co-efficient.

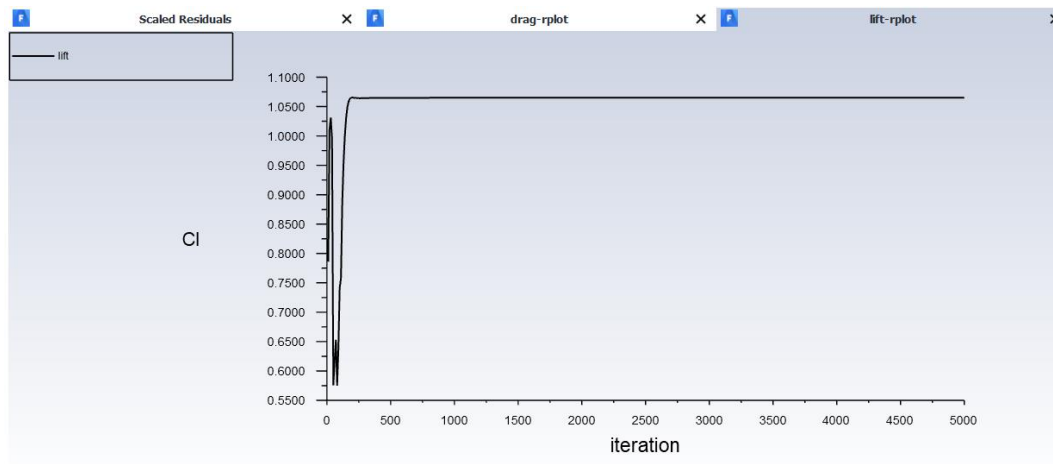


Figure 19: Lift co-efficient.

Cd	()
airfoil	0.014393384

Figure 20:  $C_d$  computed.

Cl	()
airfoil	1.0652755

Figure 21:  $C_l$  computed.

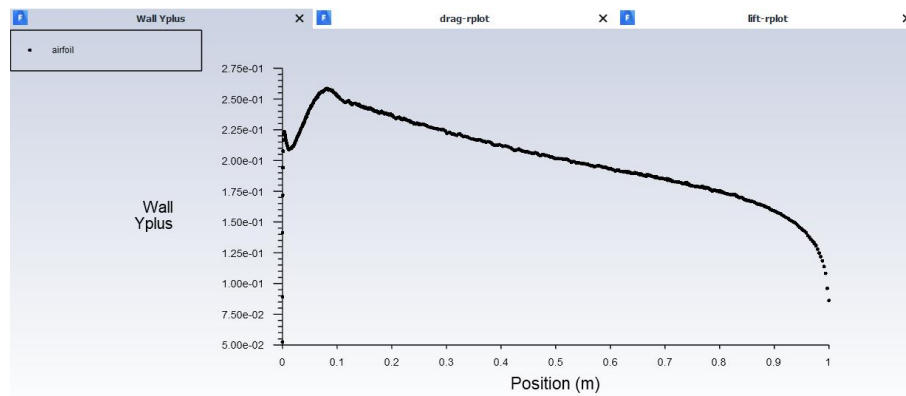


Figure 22: Wall y plus.

From the above computed values, we can see that it is almost similar to the original values from the NASA validation data table. NASA uses a high-end computer and much more chords than I used here in this simulation, because of that the values might differ a little but it is almost the acceptable value.

From the drag and lift graph we can see a slight change in the path as compared to the angle of attack of 0-degree. But it almost follows the same constant linear graph in the rest of the computation. The wall y plus is also lower than 1 which is also expected for a good simulation results.

## Conclusion

From this project the pressure and velocity gradients of different angle of attack on a NACA 0012 airfoil has been studied and the results are compared with the original NASA validation data tables which is also shown above in this report.

## References

*Airfoil Tools*. (n.d.). Retrieved from NACA 0012 Airfoils.

Rumsey, C. (2020, July 10). *NASA Turbulence Modeling Resources*. Retrieved from [https://turbmodels.larc.nasa.gov/naca0012\\_val.html](https://turbmodels.larc.nasa.gov/naca0012_val.html)

*Wikipedia*. (n.d.).