

**AEROSPACE FLUIDS ENGINEERING**  
**AER298**  
**CFD REPORT**

---

*Author*  
A.V. MYTHEN

*Registration Number*  
170154415

April 5, 2019

**ABSTRACT**

The aim of this study was to determine the relationship between the lift/drag coefficient and angle of attack for a NACA0015 wing section. This was investigated by creating a CFD model of the wing and simulating real-life boundary conditions. The data extracted from this was then validated, to some extent, by a real-life experiment using a wind tunnel. The results from both the CFD and the wind tunnel experiment suggest that an increased angle of attack lead to an increase in lift coefficient, up to the stall point (roughly 12°) and an exponential increase in the drag coefficient. Reasons for why the CFD results may be incorrect were explored.

**NOMENCLATURE**

$C_l$	Lift Coefficient	$\alpha$	Angle of Attack
$C_d$	Drag Coefficient	$\rho$	Air Density
L	Lift Force		
S	Wing Surface Area		
D	Drag Force		
V	Air Velocity		

## 1 SIMULATION PROCEDURE

In order to investigate the relationship between lift/drag coefficient and angle of attack for a NACA0015 wing section, Computational Fluid Dynamics (CFD) was used. CFD is a numerical tool which allows fluid flow systems to be simulated on a computer by solving the algebraic form of the governing differential equations of fluid flow. [1] A CFD model of the NACA0015 wing section was created using Ansys workbench. Below details the procedure that was followed to create the simulation.

### 1.1 Geometry

The geometry of the NACA0015 aerofoil must first be constructed in the software before any simulations can be run. In order to create a fluid flow simulation from the geometry creation a single system of components is used. The Analysis type for this geometry should be changed from 3D to 2D as the 2D Navier-Stokes equation should be used to simulate fluid flow, therefore the software needs to understand that data should only be stored in a 2-dimensional plane. The bottom up approach was used to create the 2-dimensional surface of the aerofoil. This approach is where points are connected together to create 1-dimensional lines which can then be connected to form 2-dimensional surfaces. To create the one-dimensional line body of the aerofoil, data points were generated from <http://airfoiltools.com/airfoil/naca4digit> [2] and downloaded onto the DesignModeler from MOLE. This 1-dimensional outline of the can then be turned into a 2-dimensional surface object using the Surfaces From Edges feature. Finally, the size of the aerofoil needs to be scaled to match the size of the model used in the wind tunnel. The length of the wing should be reduced from 1 meter to 61.5mm.

Next is to create the geometry of the environment around the aerofoil, so that we can then create a mesh from it. This is because in CFD it is the fluid, therefore the air flowing around the wing, that should be modelled, rather than the solid, in this case the wing. For 2D aerofoil studies it is common to use a 'C mesh', because the curved surface at the front of the domain allows the angle of attack to be changed by changing the angle at which air hits the wing, therefore the geometry of this C shape must be created. This is done by sketching a rectangle with vertical edges set to 1000mm and horizontal edges set to 50mm. The midpoint of the left vertical edge of the rectangle meets the trailing edge of the aerofoil. An arc is sketched so that the diameter meets the left vertical edge of the rectangle. The aerofoil surface is subtracted from the C mesh surface to leave a void. The C shape can then be split into 4 quadrants, with the trailing edge of the aerofoil as the centre point. This creates lines on which we can apply mesh sizing controls.

### 1.2 Mesh (Meshing)

Meshing is chopping up a geometry into a number of small sections (element). The finer the mesh the greater the accuracy in resolving the flow of physics due to more equations being solved, however this means a greater computational expense. Therefore, a good CFD model should have much finer mesh in areas of interest, in this case being close to the walls, and a larger mesh further away from the walls, where accuracy of the physics is less important. This can be done adjusting the number of divisions (elements) and the bias factor of the four inner lines of the quadrant, the curved line at the front which is split into two, the horizontal lines above and below and the back edge of the quadrant which is also split into two. The number of divisions is the number of elements the line gets chopped into. Initially the number of divisions on the straight lines were set to 50 and on the curved sections set to 100, however to get a more accurate resolution these were adjusted to 60 and 120 respectively. The bias factor dictates how bunched up the mesh elements are to one end. This was initially set to 10, however adjusted to 12 and set so that the elements were more bunched up closer to the aerofoil walls. This is to achieve a higher resolution on the areas of interest. There was no bias on the curved edges, only the straight lines.

Next the behaviour of each edge could be set to either soft or hard. Hard behaviour was set to the curved edge and central quadrant edges where the absolute sizing control was required and soft behaviour was applied on the external downstream edges as soft allows the mesh a bit more freedom and the cell size is not as critical here. The “quality” of the mesh can be analysed by checking the skewness using the metrics mesh tool. Once this shows the mesh is overall OK, the final step is naming the external surfaces to which the boundary conditions will be applied when imported into fluent. The two curved edges and two horizontal edges at the top and bottom of the domain are to be labelled “Inlet” and the back edge “Outlet”, while the top edge of the aerofoil is to be labelled “wall-foil top” and the bottom edge of the aerofoil “wall- foil bottom”.

### 1.3 Solve (Fluent) Setup

The next step in the process is to set up the physics of the problem so that it can be solved. Double precision, parallel and 2 processors are selected so that 2 of the machine’s cores are being used to process the solution, therefore speeding up the calculations. To ensure there are no major issues with the Mesh that has been imported an internal check is ran. Once this indicates everything is OK, the condition of Pressure-Based and Steady need to be selected. Next the physical models of how the fluids flow are to be included in the calculation. Viscous – Laminar is selected at this point as it is a symmetrical aerofoil at zero angle of attack. However later when simulating angles of attack greater than zero the k-epsilon turbulence model will be used to model turbulent flow.

Cell zone and boundary conditions are next to be considered. Cell zone conditions will already be specified to air as it is the only material in this simulation. The boundary conditions must be set correctly to realistically represent the real world that is being attempted to be simulated. The inlet velocity is edited to allow the X and Y velocity to be set separately. The X velocity is then set to 10m/s. Ensure that the outlet is set as “pressure outlet” with a gauge pressure of 0Pa and the top and bottom of the aerofoil to type “wall” with the default settings. Next set the Absolute Criteria for the residuals of the equations to be solved to 1e-06. These dictate the stopping conditions for the calculation, so the calculation will continue until each of the variables listed has a residual imbalance of 0.000001 or less.

As the equations are too complex to solve analytically, CFD uses numerical methods. In order to solve them numerically solution initialization is used to provide an initial guess. “Hybrid initialization” is used to perform a simple CFD to start the solution of the full CFD. Initialization must be done before each full CFD calculation. Calculations will run when told to either to the number of iterations specified, in this case 500 or until the convergence criteria and satisfied.

### 1.4 Results

The force exerted on the aerofoil can now be extracted and compared directly to the results obtained from the wind tunnel. In order to do so the appropriate direction vectors must be chosen. For lift, the direction vector should be  $(\sin(\alpha), \cos(\alpha))(x, y)$ , since lift is always perpendicular to the free stream velocity vector. Therefore, for an angle of attack of zero the direction vector will be (0,1). A summary of forces can be printed, and here the total net force can be found. This value will be the lift force and should be recorded into a table just like the experimental data. To get the drag force the direction vectors need to be swapped and therefore be  $(\cos(\alpha), \sin(\alpha))(x, y)$ .

In order to simulate a change in angle of attack without having to make a new geometry or remesh, the inlet velocity conditions need to be changed. The X velocity component is changed to  $V\cos(\alpha)$  (V being the velocity of the wind) and the Y component to  $V\sin(\alpha)$ . This is done for angle 0-30 degrees in increments of 5, at velocities 5.7, 8.5, 11.3, 13.5 and 16.4, just like in the wind tunnel experiment. Once all the values for lift and drag are found they must be scaled by 0.145 in order to match the span length from the experiment.

## RESULTS AND DISCUSSION

Once all the lift and drag forces were extracted, they were then used to produce lift coefficient,  $C_l$ , and drag coefficient,  $C_d$ , curves of varying angles of attack. This can be done by using equations 1 and 2 to find the lift and drag coefficients.

$$C_l = \frac{2L}{\rho V^2 S} \quad (1) \quad C_d = \frac{2D}{\rho V^2 S} \quad (2)$$

Where,  $L$  is lift force,  $D$  is Drag force,  $\rho$  is air density ( $1.225 \text{ kg/m}^3$ ),  $V$  is velocity of the air and  $S$  is the surface area of the wing, which can be found using equation 3.

$$S = \text{Chord length} \times \text{Wing span} = 0.0165 \times 0.145 = 0.0089175 \text{ m}^2 \quad (3)$$

These values can then be plotted against angle of attack and therefore graphically show the relationship between the two, as shown in figures 2 and 4. However, these graphs alone can't be relied upon as we don't know if the CFD results are correct, in that the system simulated in the computer behaves the same as the same system would in the real world. This is why the wind tunnel experiment was conducted, in order to compare these results to the CFD ones. This is known as validation.

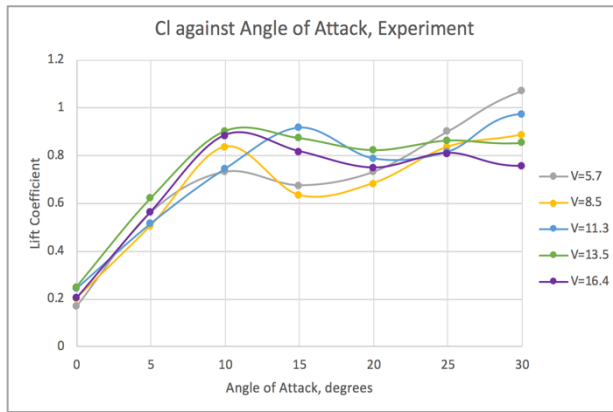


Figure 1: Graph showing  $C_l$  v AOA from the experimental data

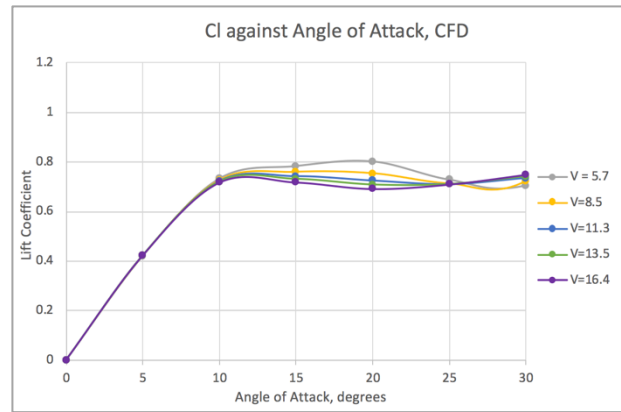


Figure 2: Graph showing  $C_l$  v AOA from the CFD data

As shown in figures 1 and 2, in both the experimental data and the CFD data the coefficient of lift increases linearly with angle of attack up to between 10 and 15 degrees, then decreases slightly and flattens out. This is where we can deduce the wing section stalls. In figures 3 and 4, it can be seen that the coefficient of drag increases with angle of attack exponentially.

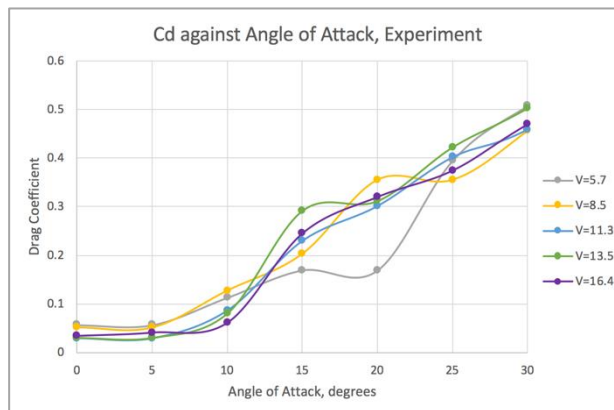


Figure 3: Graph showing  $C_d$  v AOA from the experimental data

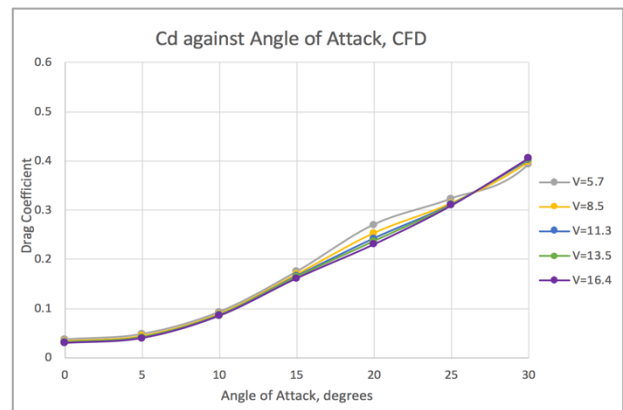


Figure 4: Graph showing  $C_d$  v AOA from the CFD data

The graphs show similar shapes and therefore a similar relationship between  $C_l/C_d$  and angle of attack. So, in terms of the relationship, the CFD results produced can be validated by the experimental

data. However, there is a fair amount of uncertainty in the experimental results therefore may not be the most reliable and should be validated themselves. As discussed in the experimental plan, some steps were taken in order to reduce uncertainties, like calibrating the equipment, however there was not enough time in the laboratory to conduct repeat readings as suggested. It can be seen that in the experimental data that the slower wind speeds produced much more fluctuating results, while at higher speeds the results were much smoother are more like the CFD. This could suggest that even cleaner results could be found at higher speeds however due to health and safety issues higher wind speeds were unable to be produced.

Although the overall relationship is the same in the CFD results as they are in the experimental results, there is still quite a bit of variation between them. There are several reasons this could be the case. In the process of setting up the CFD, many decisions have to be made in order to make the physical model as accurate as the real-world system. If some of these decisions were made poorly than it will impact the accuracy of the results as the simulation won't be as close to the real-world system as it could be. Another reason for incorrect results in CFD may be that the numerical method is not solving the equations precisely enough. CFD also has the limitation that it cannot model skin friction.

While running the calculations in CFD, for angles greater than zero, the K-epsilon model was used, as it is a very good general model for turbulent flow, however other models are available in CFD which could be argued better suited for different models. For example, the k-omega model is meant to be good as it is quite robust to near wall conditions and transitional flow [3] and the Spalart-Allmaras model was specifically designed for aerospace applications [4] therefore would suggest it's the best one for this case. However, when used in the CFD the results weren't as close to the experimental ones as the K-epsilon model.

## CONCLUSION

In this study a relationship was able to be determined between the lift/drag coefficients and angle of attack of a NACA0015 wing section by running simulation in CFD and conducting a real-life experiment using a wind tunnel. The angles of attack tested were between  $0^\circ$  and  $30^\circ$ , every  $5^\circ$  and each one was tested at 5 different wind speeds to produce a wide range of data.

The results from both CFD and the experiment showed that as the angle of attack increases, the coefficient of lift increases until the point at which the wing section stalls (roughly  $12^\circ$ ) and the coefficient of drag increases exponentially. Although the relationships are the same for the experimental data and the CFD data, some cautious scepticism should be considered while interpreting any CFD results. As discussed in the discussion and results, validation is always needed when using computers to simulate real-life models. In order to make the results of this study more valid, repeat readings should have been conducted in the experiment. Also the experimental data could have been validated from published studies.

## REFERENCES

- [1] Dr Andrew Garrard MEng PhD SFHEA, *A Crash Course in CFD*, [Reviewed April 2019]  
[https://vle.shef.ac.uk/bbcswebdav/pid-3711924-dt-content-rid-18022217\\_1/courses/AER298.A.199395/p\\_cfdintro\\_pics.pdf](https://vle.shef.ac.uk/bbcswebdav/pid-3711924-dt-content-rid-18022217_1/courses/AER298.A.199395/p_cfdintro_pics.pdf)
- [2] NACA 4 Digital Airfoil Generator, [Reviewed April 2019]  
<http://airfoiltools.com/airfoil/naca4digit>
- [3] Mole, *Tips for completing the AER298 Assignment*, [Reviewed April 2019]
- [4] Wikipedia, *Spalart-Allmaras Turbulence Model*, [Reviewed April 2019]  
[https://en.wikipedia.org/wiki/Spalart%E2%80%93Allmaras\\_turbulence\\_model](https://en.wikipedia.org/wiki/Spalart%E2%80%93Allmaras_turbulence_model)