Analysis of an airfoil
using Computational Fluid Dynamics

Tanveer Chandok
12/17/2010

Independent research thesis at the Georgia Institute of Technology under the supervision of Professor Lakshmi Sankar
# Table of Contents

Introduction........................................................................................................................................... 2  
Description of Tools ............................................................................................................................... 3  
Inputs and Constraints........................................................................................................................... 4  
Procedure .................................................................................................................................................. 5  
Results and Discussion .......................................................................................................................... 10  
Conclusions and Recommendations ...................................................................................................... 17  
References............................................................................................................................................... 19  
Appendices................................................................................................................................................ 20  
  Appendix I: Acronyms used .................................................................................................................. 20

**Note: All acronyms used are defined in Appendix I**
Introduction

A wide range of engineering devices employ airfoils operating at relatively low speeds. The analyses of the performance of airfoils at these low speeds helps designers make accurate and informed decisions when it comes to choosing an airfoil for a task.

The airfoil that will be analyzed in this report will be the NACA 0018. This airfoil has a rich history as it was used on the famous Lockheed-Vega 17 B-17 Flying Fortress as well as other airplanes in World War II\(^1\). There are various constraints and assumptions that will be used. The freestream velocity, angle of attack, pressure, density, temperature and kinematic viscosity will all be kept constant. Using two popular software (in addition to other basic tools), a structured mesh will be constructed and using 2D airfoil analysis, the lift coefficient, velocity vectors, and pressure contours will be plotted and studied. The data obtained will be compared to reliable sources to check for accuracy and errors if any.

\(^1\) The Incomplete Guide to Airfoil Usage, David Lednicer, 2010, University of Illinois at Urbana Champagne: http://www.ae.illinois.edu/m-selig/ads/aircraft.html
Description of Tools

In order to analyze the airfoil, it must first be drawn in CFD meshing software. The meshing software used was Gridgen, by Pointwise. This software has been used since 1984. The software generates structured hexagonal, unstructured tetragonal and hybrid meshes. The meshes it generates are of high quality, leading to more accurate solutions and faster convergence\(^2\). In order to draw the grid of the NACA 0018 airfoil, data points are required. These data points can be obtained from an online java applet called JavaFoil\(^3\). Figure 1 (below) shows a screen shot of the applet being used to find the data points of the required airfoil. This software is free to use and can generate almost any standard airfoil being used today. It offers various capabilities such as specification of thickness, thickness location, camber and camber location. The number of data points required can also be changed.

Once an airfoil has been set up and meshed, it needs to be analyzed in a flow modeling software. The program used in this research paper was FLUENT, by ANSYS. This software contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfers, and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace. This software has an advanced solver technology that provides fast, accurate CFD results, flexible moving and deforming meshes, and superior parallel scalability\(^4\). It can also readily import the meshes created in Gridgen and hence the cross compatibility ensures conserved data.

Using the tools described above, the 2D airfoil can be analyzed efficiently.

\(^3\) JavaFoil, Martin Hepperle, 1996-2008: http://www.mh-aerotools.de/airfoils/jf_applet.htm
Inputs and Constraints

The mesh that will be created prior to the calculations will be on a structured domain.

In Fluent, standard sea level values will be assumed for the freestream properties. Hence,

Pressure = 101,325 Pa
Density = 1.2250 kg/m³
Temperature = 288.16 K
Kinematic viscosity = 1.407e-5 m²/s

The freestream velocity will be 50 m/s and the angle of attack 2°. The viscous model will be *inviscid* and the flow will be considered incompressible as the Mach number used is around 0.15.
Procedure

The first step to analyze an airfoil requires the cross section (in 2D) to be converted into data points that can be graphed. To find these data points, JavaFoil was used (as shown below in Figure 1).

The NACA 0018 airfoil is a generic NACA airfoil with 18% thickness at 30% of its chord length. By inputting these values into the applet, the correct airfoil was obtained. The data points obtained were copied into a text file and were used to generate the airfoil in the meshing software.
The data point text file was inputted into Gridgen using the “Database Import” function. The next step was to define connectors and the distance between them. This helps in drawing a mesh around the airfoil that can later be manipulated.

Figure 2: Airfoil structure after importing the database of points

While creating connectors, one of the connectors was created 11 times the chord length behind the airfoil. The airfoil length is one unit (1.), so entering 11 for the length of the wake connector puts the exit boundary at 10 body lengths downstream. This distance is a typical value for many analysis problems. The points along the connectors (where the mesh is generated from) need to be redistributed so that they are all equally spaced (the function ReDistribute is used for this).

The final step was to actually create the mesh (using Domain Extrusion). For the purposes of the analysis required, a structures mesh was generated.

---

After defining the region of the domain, the size of the mesh was defined. The boundary condition used was 10 times the total height of the airfoil. Since the wake connector was 10 body lengths away, this height of 10 body lengths would be adequate. Figure 4 (next page) shows the final extruded structured mesh airfoil that was ready for analysis in Fluent.
Figure 4: Final structured mesh of the NACA 0018 airfoil

The actual analysis takes place is Fluent. There are many versions of Fluent but the one that was used in this experiment was “2ddp”. The cross compatibility between Gridgen and Fluent was a great help in importing/exporting the mesh. The first step was to set up the experiment parameters in Fluent. The analysis was pressure based, not density based. The boundary conditions needs needed to be modified to make sure that the different parts of the mesh corresponded to their “real-world” counterparts. While defining the “Velocity Inlet”, the
angle of attack is defined as 1.2° instead of 2° (as mentioned earlier) in order to adjust for any error caused by assuming the airfoil to be 2D instead of 3D. The gauge pressure at the pressure outlet was set to 0 at the boundary. All the other parameters are listed in the Inputs and Constraints section (page: 4).

While running the solver, the pressure was set to the “PRESTO!” default setting and momentum to “Second-Order Upwind”. Next, the different force vectors were defined for lift and drag and their initial conditions were assigned.
Results and Discussion

After solving, the solution converged after 190 iterations (as shown in Figure 5 below).

Fluent gives us the continuity, x-velocity, y-velocity, coefficient of lift, coefficient of drag, the time it took to converge the solution.

<table>
<thead>
<tr>
<th>Iter</th>
<th>Continuity</th>
<th>X-Velocity</th>
<th>Y-Velocity</th>
<th>Cl</th>
<th>Cd</th>
<th>Time/Iter</th>
</tr>
</thead>
<tbody>
<tr>
<td>188</td>
<td>1.0888e-06</td>
<td>1.9312e-08</td>
<td>7.2772e-09</td>
<td>1.3215e-01</td>
<td>4.6470e-03</td>
<td>0:00:18:081</td>
</tr>
<tr>
<td>189</td>
<td>1.0695e-06</td>
<td>1.8450e-08</td>
<td>6.9401e-09</td>
<td>1.3215e-01</td>
<td>4.6470e-03</td>
<td>0:00:14:082</td>
</tr>
</tbody>
</table>

Figure 5: Solution converged at 190 iterations

Note: The coefficient of drag is expected to be very close to zero (inviscid vicious model is being used). When the solution converges, the true values of the airfoils performance is revealed at the specific angle of attack, as shown in the table below:

<table>
<thead>
<tr>
<th>Coefficient of Lift (C_l)</th>
<th>0.13215</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coefficient of Drag (C_d)</td>
<td>0.00464</td>
</tr>
</tbody>
</table>

Table 1: C_l and C_d

The corresponding graphs on the next page show scaled residuals (Graph 1) and the lift convergence history (Graph 2).
Graph 1: Scaled Residuals

Graph 2: Lift convergence history
Using Fluent's various capabilities, the following plots were generated.

**Plot 1:** Contours of Pressure Coefficient (in pascal's) over the entire airfoil

**Plot 2:** Contours of Velocity Magnitude (in m/s) over the entire airfoil
Plot 2: Contours of Velocity Magnitude (in m/s), focused on LE

Plot 3: Contours of Velocity Magnitude (in m/s), focused on TE
**Plot 4:** Velocity vectors over the entire airfoil colored by velocity magnitude (in m/s)

**Plot 5:** Velocity vectors focused on the LE colored by velocity magnitude (in m/s)
Plot 6: Velocity vectors focused on TE colored by velocity magnitude (in m/s)

Plot 7: Pressure Coefficient over the entire airfoil

From the plots shown, the airfoils aerodynamic properties can be analyzed. Since this airfoil is symmetrical, and is operating at a very small angle of attack, the velocity of the upper surface is almost the same as the velocity on the lower surface. From Plot 2 it can be seen that
the velocity just behind the LE is much greater than that at the LE. In Plot 5 there is a stagnation point on the LE where the velocity flow is nearly zero (also seen in Plot 2). The fluid (air) accelerates on the upper and lower surfaces as can be seen from the change in colors of the vectors. In Plot 6 and Plot 3, it can seen that the flow on the upper and lower surfaces decelerate and converge together. Plot 1 show’s that there is a region of high pressure at the LE (stagnation point) and region of low pressure on the upper and lower surfaces of the airfoil (Lower pressure on upper surface). This is what was expected from analysis of the velocity vector plot. From the Bernoulli equation, it is known that whenever there is high velocity, there is low pressure, and vice versa. In Plot 7, the lower curve is the upper surface of the airfoil and has a negative pressure coefficient as the pressure is lower than the reference pressure.
Conclusions and Recommendations

The lift coefficient for inviscid model is higher than the experimental value. In reality, if the effect of viscosity was taken into consideration, \((C_l)_{\text{skin friction}}\) would have a negative value and hence reduce the coefficient of lift according to the following equation:

\[
C_l = (C_l)_{\text{pressure}} + (C_l)_{\text{skin friction}}
\]

The value obtained for the Coefficient of Lift is the very close to various other online sources\(^6\).

The \(C_d\) was predicted to be 0, since the analysis took place in an inviscid environment. The \(C_d\) obtained was 0.00464. This is very close to 0, but there is a slight error. This error can be attributed to the fact that the angle of attack was not zero. At 2° angle of attack, the airfoil itself will generate slight drag. This is a common occurrence. However, the drag generated is too small to make any significant change to the airfoil’s performance.

This report shows that in order to test the performance of an airfoil, there are various conditions that need to be known (or assumed). Sometimes, these assumptions do not work well and must be changed. The angle of attack had to be changed to 1.2° as this was a 2D analysis. This 60% change is necessary to get values that are close to their actual ones.

The programs used were adequate for this type of analysis. There were various functions that were not used since the aim of this report was to only touch the surface of CFD. These programs can be (and are being) used for complex airfoil analysis in 2D and 3D. As shown in this report, everything about an airfoil can be accurately described using the tools provided. All of the conclusions from the plots drawn correspond to theoretical data.

\(^6\) Airfoil Investigation Database, 2010: http://www.worldofkrauss.com/foils/1240
CFD is a vast field that is gaining popularity as it enables engineers to accurately and efficiently analyze components. What took weeks to do in the past can now be done in a few hours. This field has great potential and is rapidly growing with new software entering the market yearly. In the future, CFD will solve problems that have not even been recognized today.
References


- **Pointwise** Inc, 2010: http://www.pointwise.com/gridgen/


- **Pointwise Gridgen Tutorial**, 1997-2003:


- **Cornell University**, SimCafe, 2009:
  https://confluence.cornell.edu/display/SIMULATION/FLUENT++Flow+over+an+Airfoil+-+Step+4
Appendices

Appendix I: Acronyms used

CFD: Computational Fluid Dynamics

Pa: Pascals

kg: Kilogram

m: meters

K: Kelvin

s: second

\( \nu \): Kinematic viscosity

\( C_l \): Coefficient of Lift

\( C_d \): Coefficient of Drag

LE: Leading Edge

TE: Trailing Edge