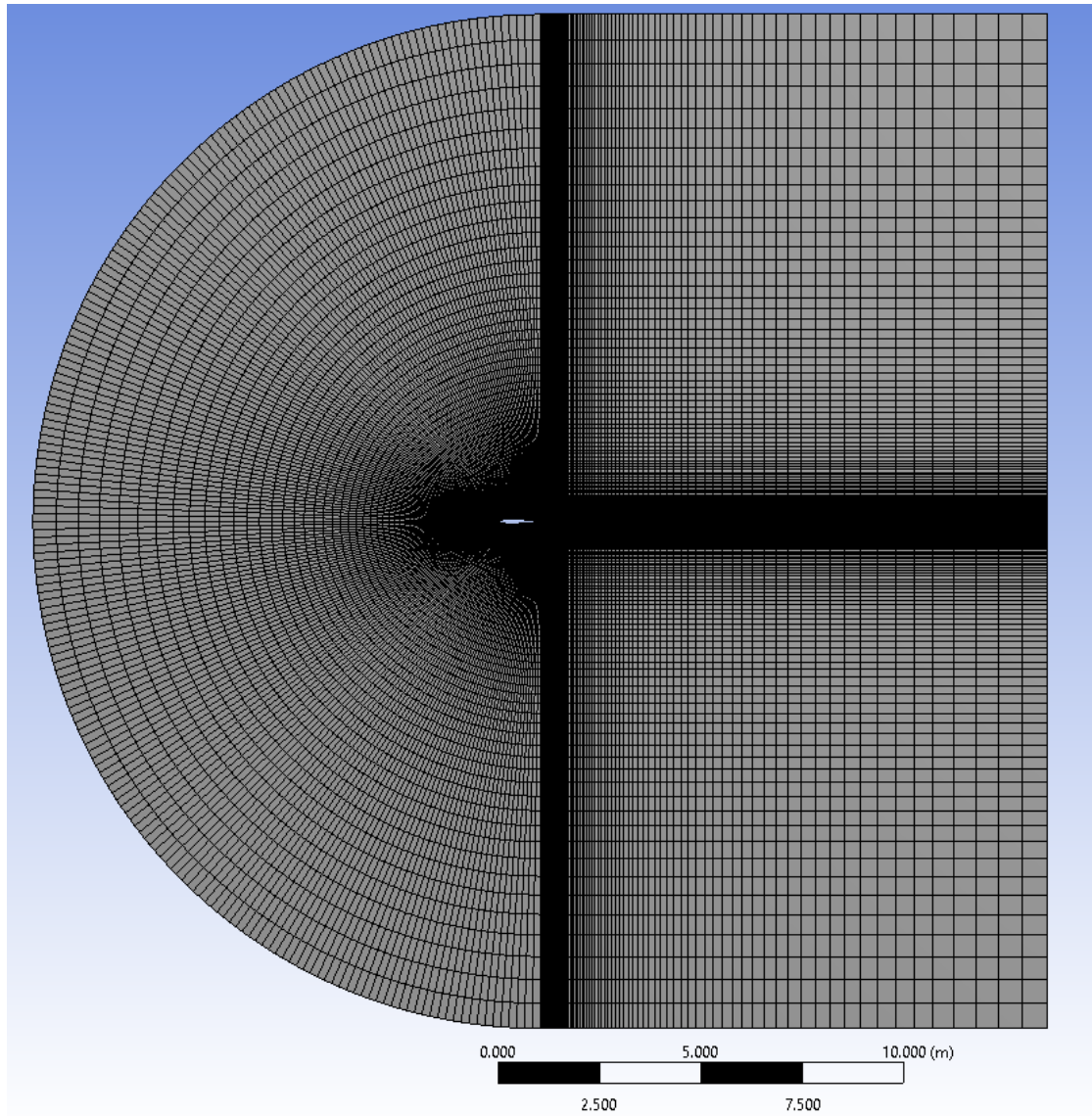


# ME 356: Fluid Mechanics Project – Spring 2018



Professor: Dr. Zhexuan Wang

Student: Yusuf Wong

May 28, 2018

## **Abstract:**

Using ANSYS FLUENT Version 18.2 software, data was obtained while inputting an inlet velocity of 0.5 m/s at an attack angle of  $11^\circ$  at steady, irrotational, incompressible, and inviscid flow. We were able to obtain lift and drag coefficients for the NACA 0012 airfoil at a Reynolds number (Re) of  $2.88 \times 10^6$ . The fluid flow was air at a density of  $1 \text{ kg/m}^3$ . By using a mesh of 15,000 elements, convergence was found after 6778 iterations, and by using a mesh of 40,000 elements, convergence was found after 3772 iterations. The drag and lift coefficients for the 15k mesh were 0.024959401 and 0.86168835, respectively. For the 40k mesh, drag and lift were 0.01281279 and 1.1613223, respectively. Additionally, contour graphs of pressure and velocity were obtained.

## **Introduction:**

For this software experiment, data was collected in order to show the effects of an air flow over a specific body. A C-mesh was created in this software that represents the control volume of the air flowing on the surface of the body, as shown in Figure 1. The NACA 0012 airfoil was used to represent this body, as shown in Figure 1. The initial conditions for this experiment had an inlet velocity of 0.5 m/s at an attack angle of  $11^\circ$  with respect to the horizontal. This fluid is steady, irrotational, incompressible, and inviscid.

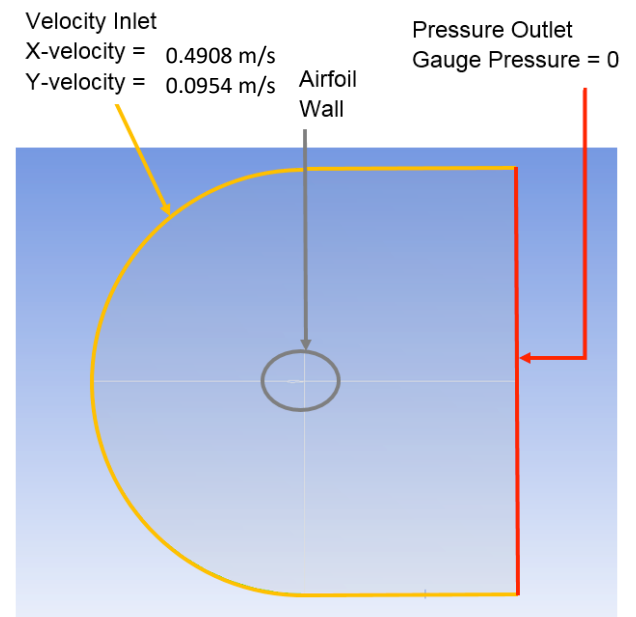


Figure 1: C-Mesh Diagram

The drag force coefficient is expected to be 0 because of D'Alembert's paradox. He states that the drag force on any nonlifting body of any shape immersed in a uniform stream is 0 when using the irrotational flow approximation. Because air is flowing over this airfoil body and the flow is irrotational, the drag force is 0 since we assume that the pressure at the front of the airfoil is equal to the pressure at the back of the airfoil by using the irrotational flow approximation. However, this is not true; the drag coefficient is not 0 based on experimental calculation with fluent. This is the reason why Alembert's theory is a paradox.

### **Assumptions:**

For this experiment, the flow is steady, inviscid (which implies no friction), incompressible, and irrotational. Because the flow is inviscid, the velocity on the edge of the airfoil is not 0; this is a no no-slip condition case, which implies that the velocity across the edge of the airfoil is the streamline velocity. Also, because the flow is inviscid,  $\mu = 0$  in the Navier-Stokes equation. Gravity is neglected in this simulation as well. The flow is 2-dimensional (XY axis). There is no z-component of velocity. As a result, this simplifies and reduces the Continuity and Navier-Stokes Equations, as shown in the following section.

### **Governing Equations:**

Continuity:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

Navier-Stokes:

$$\rho \left( \frac{u \partial u}{\partial x} + \frac{v \partial u}{\partial y} \right) = - \frac{\partial P}{\partial x} \quad (\text{x-component})$$

$$\rho \left( \frac{u \partial v}{\partial x} + \frac{v \partial v}{\partial y} \right) = - \frac{\partial P}{\partial y} \quad (\text{y-component})$$

Drag Coefficient

$$C_D = \frac{F_D}{\frac{1}{2} \rho V^2 A}$$

Lift Coefficient

$$C_L = \frac{F_L}{\frac{1}{2} \rho V^2 A}$$

### **Methods (Fluent Setup & Procedure)**

To start this experiment, a C-mesh was created around the NACA 0012 airfoil; the C-mesh is like a small bullet with a huge radius of 12.5 meters and a total length of 25 meters. This C-mesh is the control volume of the inviscid air flow, and the airfoil is basically what the air is attacking once it

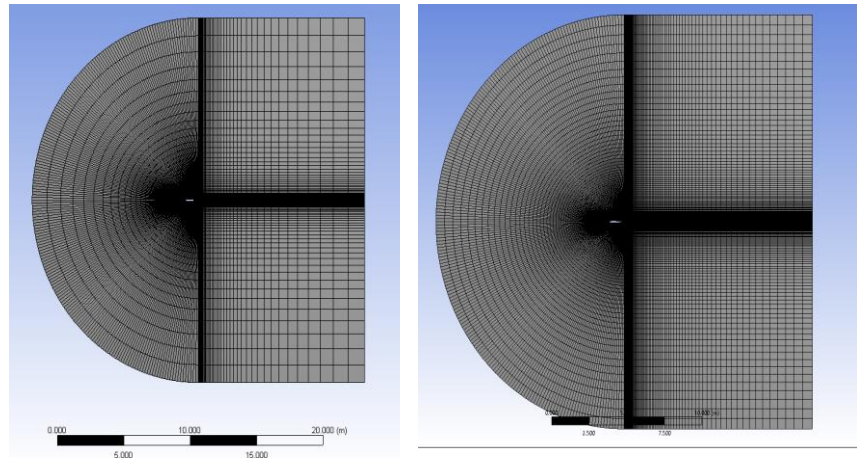


Figure 2: 15k Mesh (left) & 40k Mesh (Right)

hits the airfoil; one can think the air foil as the outline of a wing and the C-mesh as air hitting the wing at a certain velocity and attack angle. Once the geometry was made, 2 simulations of mesh size were made: 15,000 elements and 40,000 elements, as shown in Figure 2. The purpose of this experiment was to also see the differences in values in a more accurate mesh vs. a less accurate mesh.

The mesh was then setup according to the assigned model. The model was to be inviscid air at a density of 1 kg/m<sup>3</sup>. The inlet velocity was set to 0.5 m/s at an attack angle of 11°; the x-component was set to 0.4908 m/s and the y-component was set to 0.0954 m/s. Gauge pressure was also set to 0 Pa.

After setup, flow was set to be calculated by second order upwind in order to get more accurate results. The criteria for continuity was set to  $1 \times 10^{-6}$  for the limit of convergence. This implies that the number of iteration of air flow across the air foil will not stop until the difference is as small as  $1 \times 10^{-6}$ . Finally, the maximum number of iterations set was 10,000 for both the 15k and 40k meshes to be calculated. Note that the number of iterations did not have to reach 10,000; the simulation stops until it reaches convergence of  $1 \times 10^{-6}$ .

### **Fluent Results of 15k Mesh:**

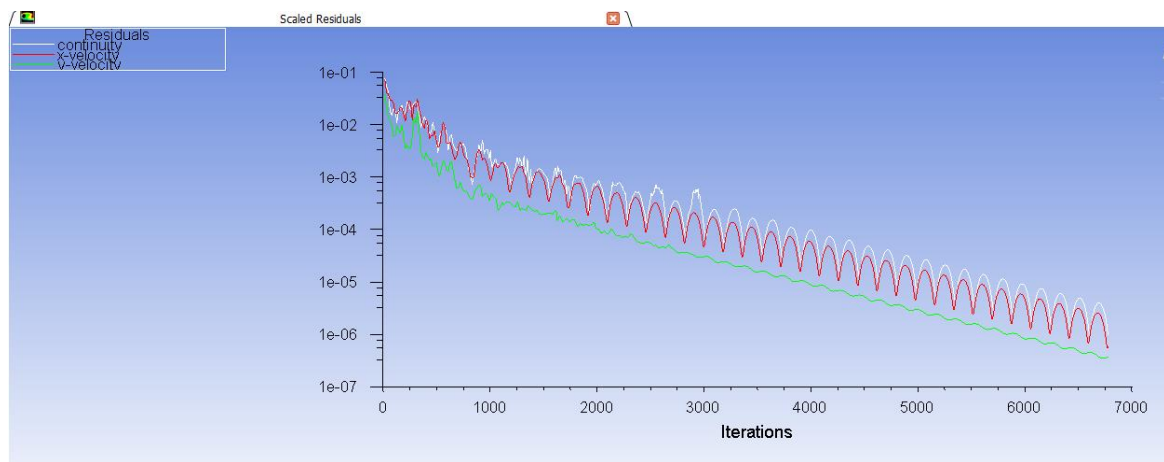


Figure 3: Convergence vs. Number of Iterations (15k Mesh)

Figure 3 shows that the 15k mesh converged at approximately 6778 iterations.

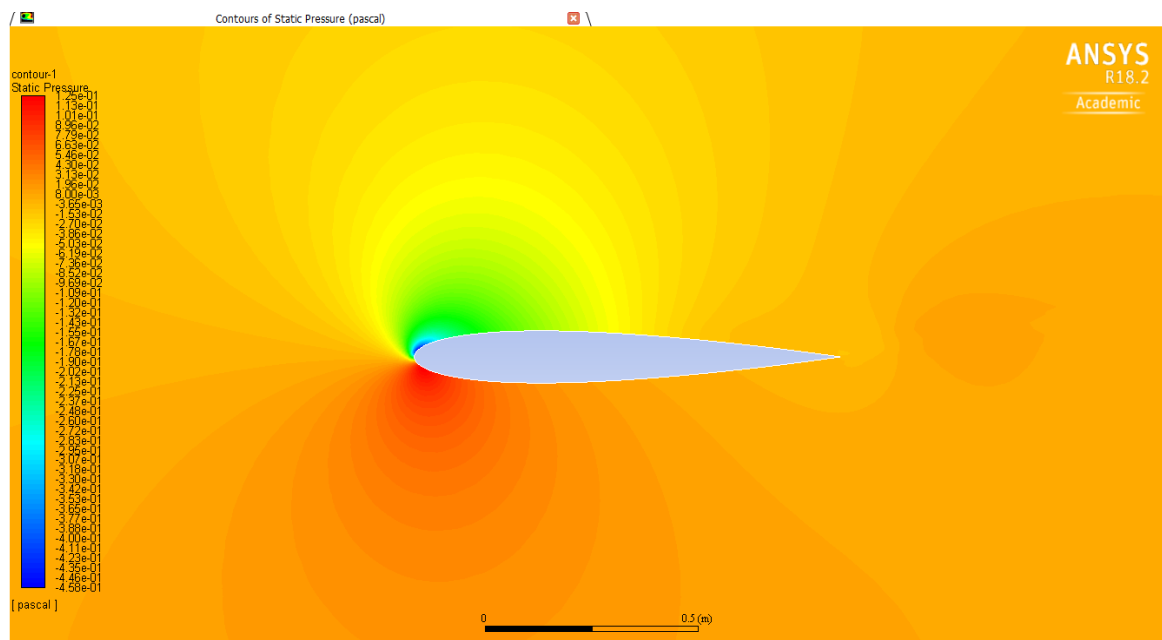


Figure 4: Contours of Static Pressure at Airfoil (15k Mesh)

Figure 4 shows that there is less static pressure at the top of the airfoil than on the top, which gives this airfoil lift, which is why the lift coefficient was calculated.

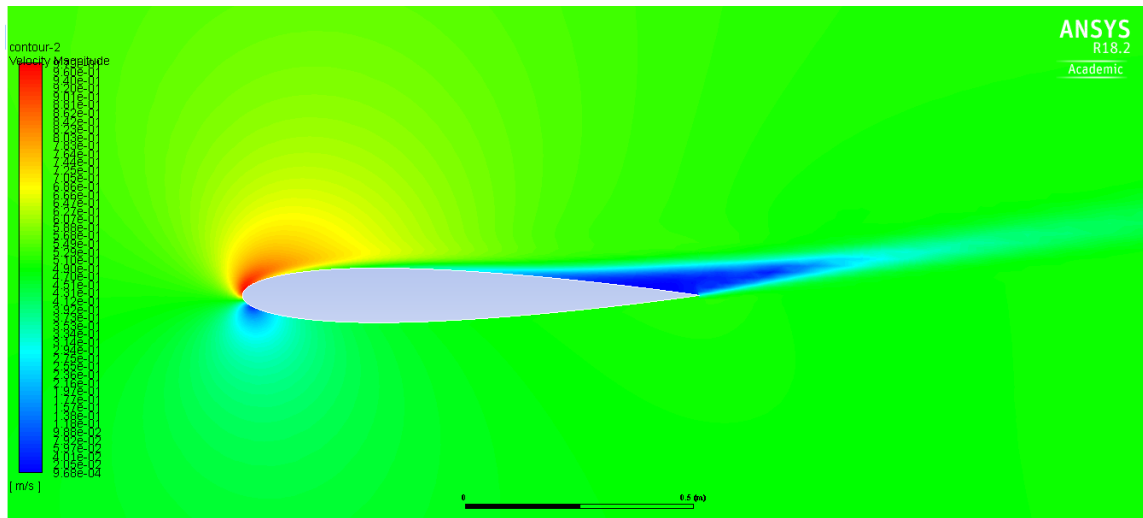


Figure 5: Contours of Velocity at Airfoil (15k Mesh)

Figure 5 shows that most of the high velocity airflows are on the top of the airfoil than at the bottom, which makes sense because the higher the velocity at the top, the less pressure there is, as shown in Figure 4 (there is less pressure on top of figure 4 than on the bottom).

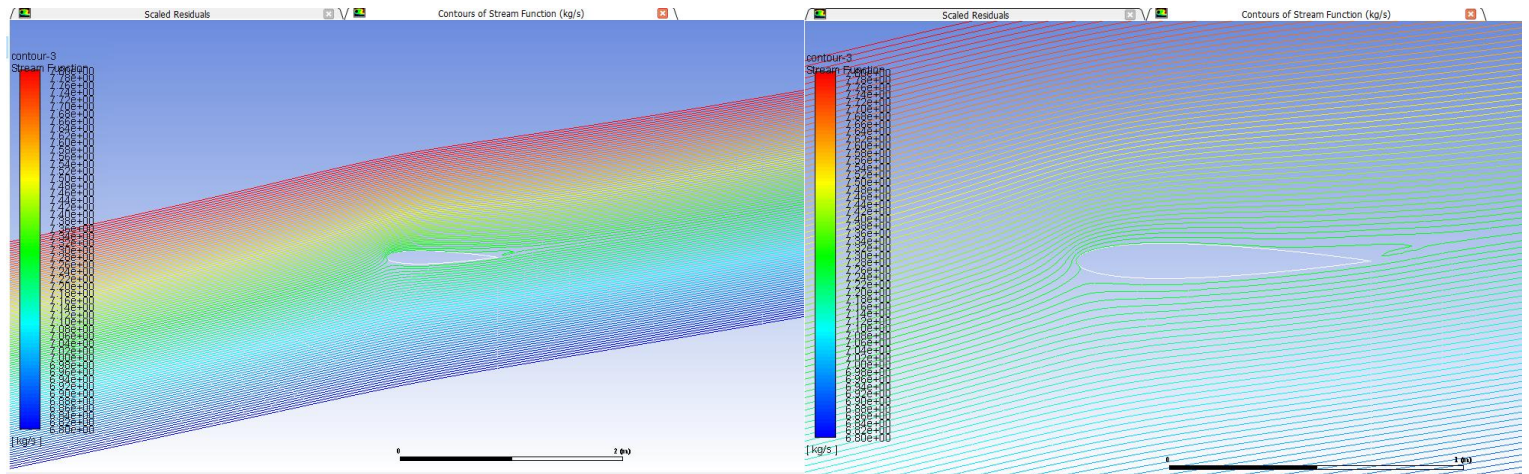


Figure 6: Contours of Stream Function (Right Pic is Zoomed In)-15k Mesh



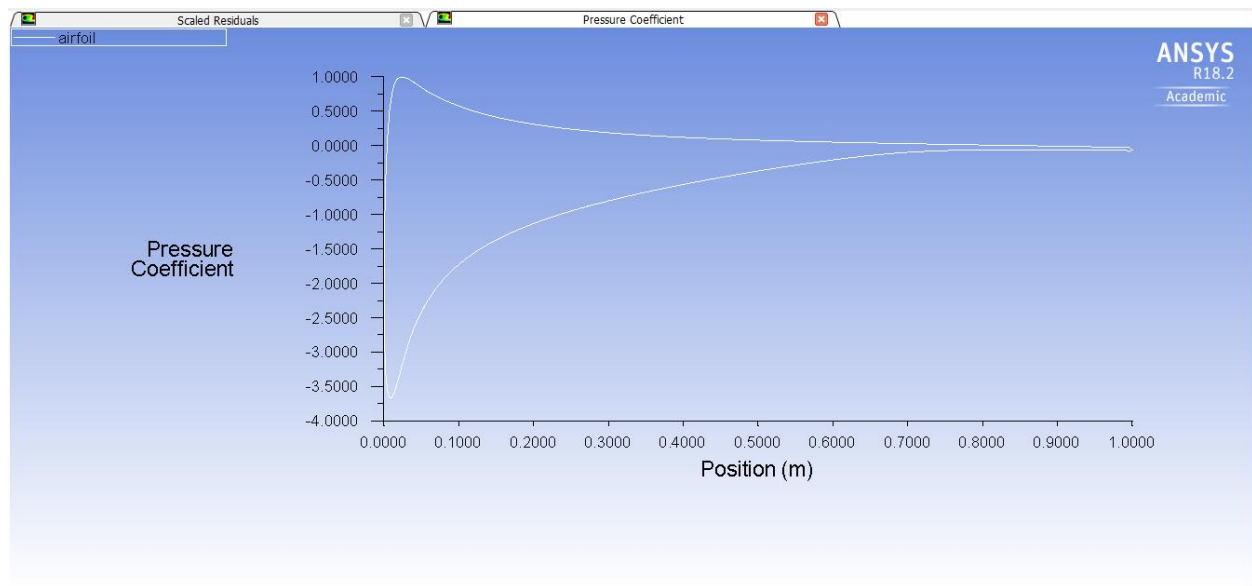


Figure 7: Pressure Coefficient vs. Position (m)-15k Mesh

### **Fluent Results for 40k Mesh:**

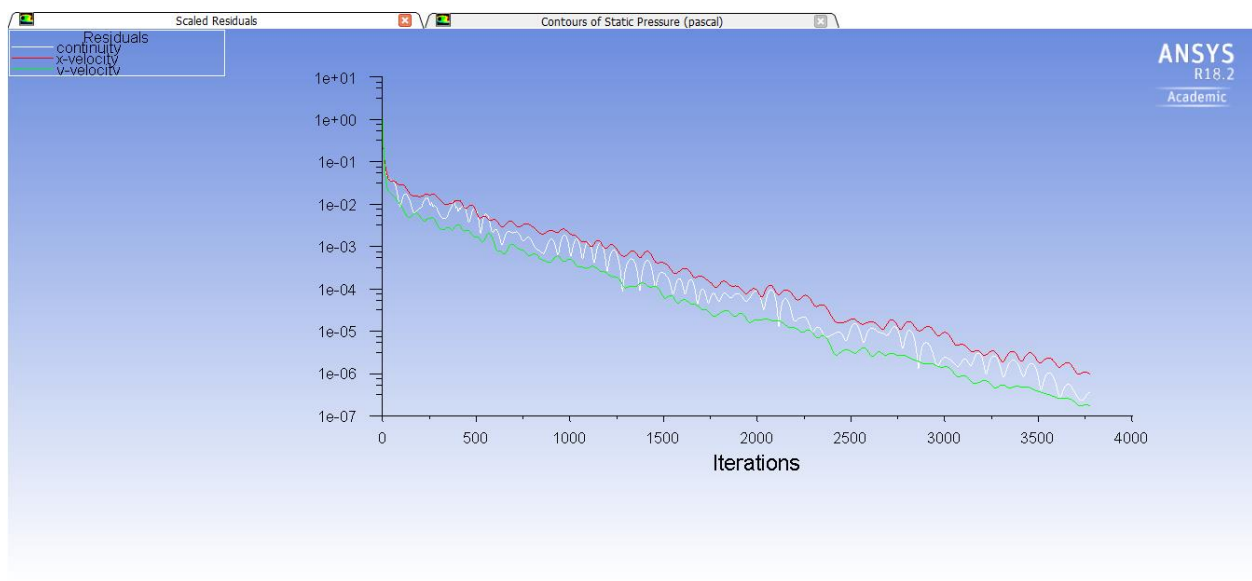


Figure 8: Convergence vs. Number of Iterations (40k Mesh)

Figure 8 shows that the 40k mesh converged at approximately 3772 iterations, which is almost twice as less as the 15k mesh.

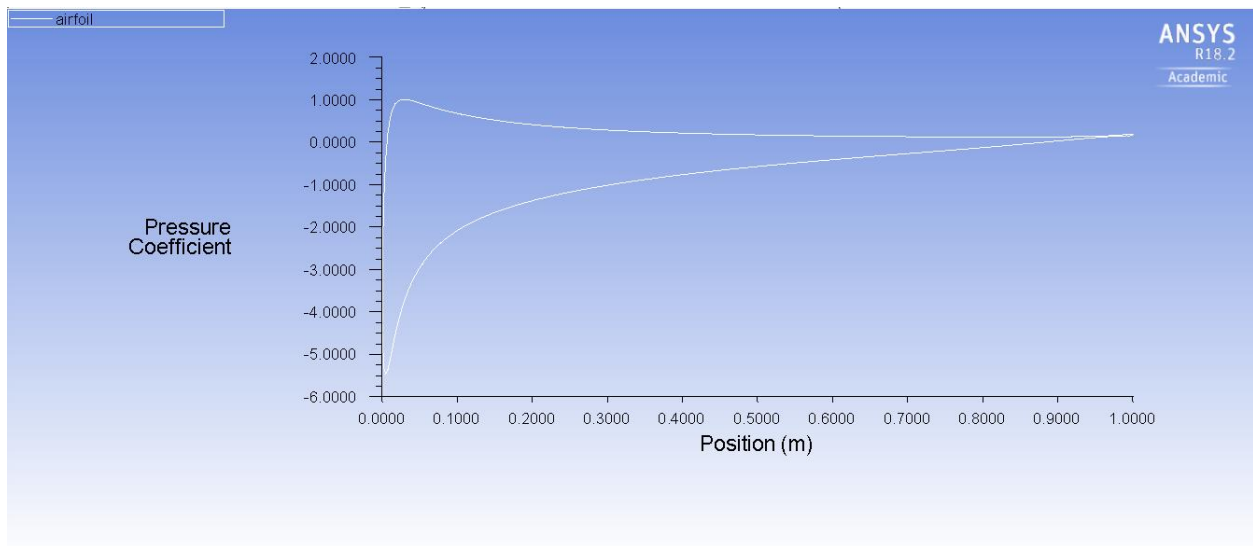


Figure 9: Pressure Coefficient vs. Position (m) (40k Mesh)

### Coefficient of Pressure Comparison:

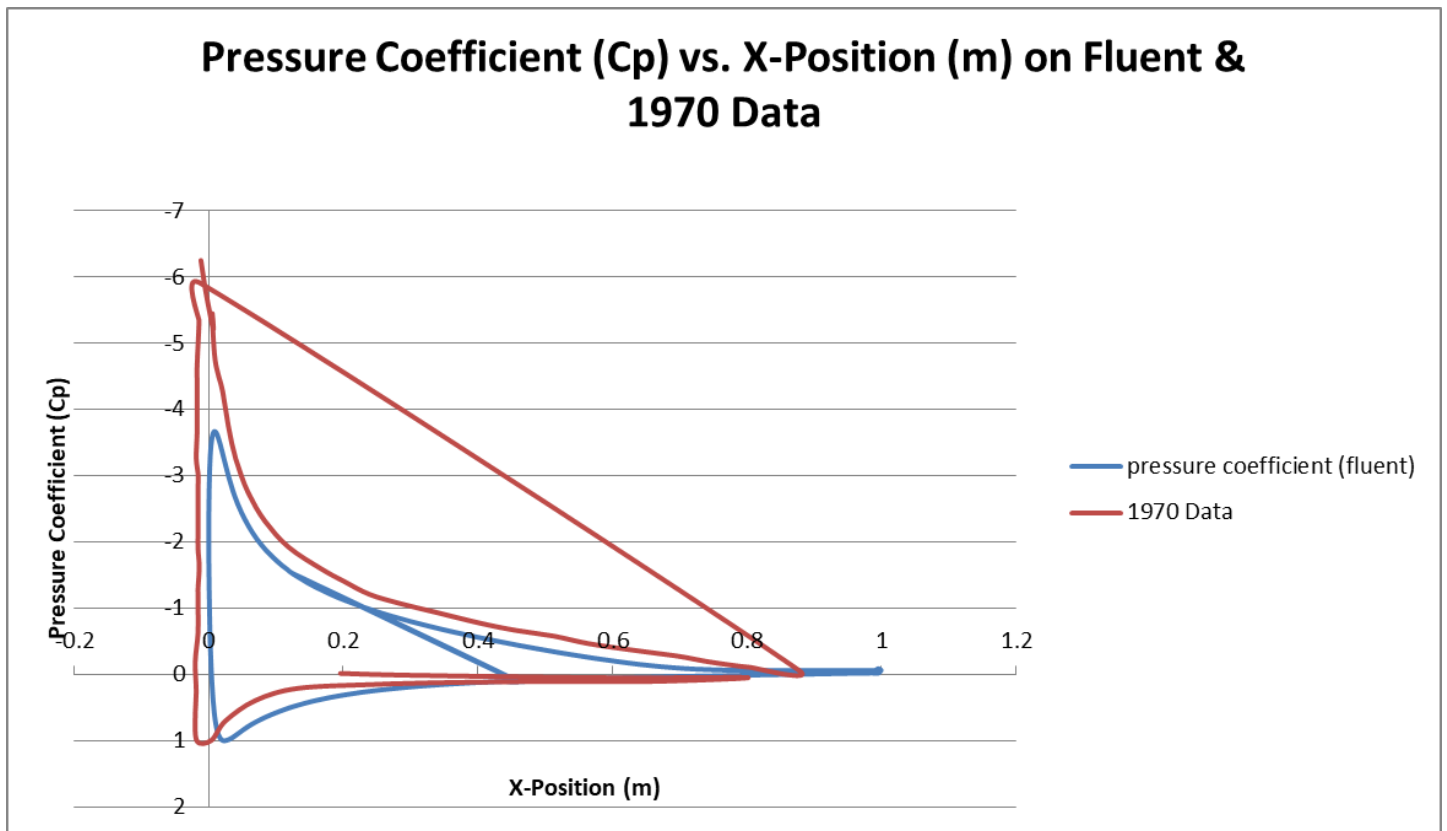


Figure 10: Pressure Coefficient (15k Mesh) vs. X-Position for Both Fluent Simulation and 1970 Experiment



### Lift and Drag Coefficient Comparison:

	15,000 Mesh (unrefined)	40,000 Mesh (Refined)	Experimental Data from Gregory & O'Reilly, NASA R&M 3726, Jan 1970
Drag Coefficient	0.024959401	0.01281279	0.015453030239365798
Lift Coefficient	0.86168835	1.1613223	1.1691877809992022

Table 1: Lift and Drag Coefficients of 15k Mesh, 40k Mesh, and Experimental Data from NASA

Table 1 shows that the experimental data is more closely represented by the 40k mesh than the 15k mesh. This make sense because the 40k mesh takes more precise positions of the air flow across the airfoil to determine the drag and lift coefficients, yielding results more similar to the 1970 experimental data than the 15k mesh.

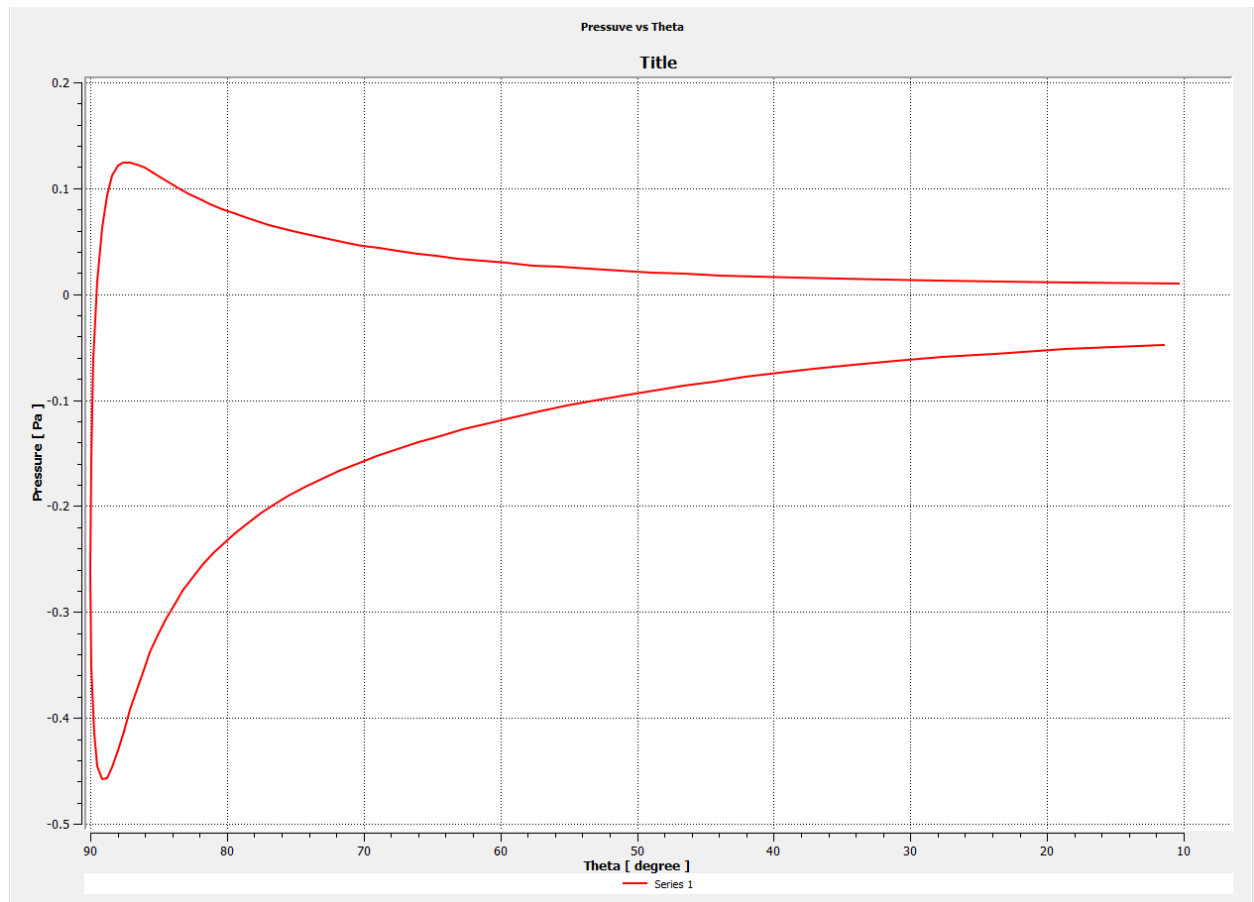


Figure 11: Pressure vs. Theta (Angle of Attack)

## **Conclusion & Final Remarks:**

As a result of this experiment, the 15k and 40k mesh show differences in values for certain graphs. For example, by having a denser mesh (40k), convergence was determined at a smaller number of iterations than the 15k mesh. Having a denser mesh also yielded different values for the drag and lift coefficients; in fact, these values were much closer to the values obtained from the 1970 NASA experiment. It seems that for this experiment, a denser mesh gave more accurate results. Additionally, the pressure coefficient vs. x-position graphs for the 15k and 40k meshes were different; the maximum and minimum coefficients were not the same. This explains why, in Figure 10, the data from the 15k mesh doesn't exactly lie on top of the data from 1970; if the pressure coefficient vs. x-position data from the 40k mesh were to be compared with the NASA data, then both data graphs would more evenly match.

Finally, Figure 11 shows two points of Pressure per angle theta; the lower point represents the pressure at the bottom of the airfoil and the upper point represents the pressure at the top of the air foil. By running the simulation, pressure increases on the bottom of the airfoil as the angle of attack becomes smaller. Likewise, pressure decreases on the top of the airfoil as the angle of the attack become smaller.

## **References**

ANSYS FLUENT 12.0/12.1 Documentation. Retrieved November 11, 2017, from

<http://www.afs.enea.it/project/neptunius/docs/fluent/index.htm>

FLUENT Learning Modules. Retrieved November 11, 2017, from

<https://confluence.cornell.edu/display/SIMULATION/FLUENT+Learning+Modules>

“Fluent 18.2 Tutorial-Case Study: Flow Over an Airfoil”. Aerodynamics Laboratory,

Department of Mechanical Engineering. City College of New York.